Innovative Tools. Excellent Service.

Batronix Magnova®
User Manual

Document Version 1.1.0 (19th June 2025)

w w w . b a t r o n i x . c o m

Table of contents

1. Introduction

Congratulations on your purchase of a Magnova oscilloscope! Oscilloscopes are indispensable tools for recording and analyzing electrical signals. They allow you to test complex circuits, troubleshoot, and make informed decisions when designing and maintaining electronic systems.

Before using your oscilloscope, it is important to familiarize yourself with basic safety precautions and regulatory requirements to ensure safe and compliant operation.

1.1. Safety instructions

The safety instructions are of crucial importance to protect both the user and the equipment from damage. There are numerous sources of danger that can occur if the oscilloscope is used improperly.

1.1.1. Power supply technology

- Use the supplied power cord or a replacement cord recommended by the manufacturer. This will ensure that the electrical specifications of the equipment are met.
- Ensure that the mains voltage matches the specifications on the device's rating plate. Incorrect voltage can cause serious damage to the oscilloscope and endanger your safety.
- Never connect the oscilloscope to an overloaded power socket to avoid the risk of an electrical fire.
- Do not use an isolating transformer to connect the appliance to the mains power supply. An isolating transformer defeats the protective grounding of the equipment and increases the risk of electric shock.

1.1.2. Grounding

- Ensure that the appliance is properly grounded to avoid electric shocks. Missing or faulty earthing can bring life-threatening voltages to the housing of the appliance.
- Connect all test leads and probes according to the instructions in the manual to ensure that no dangerous potential differences occur. Incorrect wiring can cause malfunction and safety hazards.

1.1.3. Operating environment

- Only operate the oscilloscope in a dry environment and protect it from moisture and liquids. Liquids in and on electronic components and assemblies can cause short circuits or electric shock.
- Avoid operation in dusty or explosive atmospheres. Dust can block the ventilation openings and lead to overheating, while explosive vapors could be ignited by sparks.
- Place the unit on a stable and secure surface to prevent it from tipping or falling. A fall may not only damage the equipment, but may also cause dangerous situations due to exposed electrical components.

1.1.4. Use of probes and measuring cables

- Check the condition of the measuring cables and probes regularly and replace damaged parts immediately. Damaged measuring cables and probes can transmit dangerous voltages without protection and provide incorrect measurement results.
- Only use the measuring cables and probes within the specified voltage ranges to prevent damage to the oscilloscope and danger to the user. Exceeding the voltage limits can irreparably damage the device and create dangerous situations.
- It is imperative that the input voltage at the channel inputs does not exceed the maximum value specified in the data sheet. Failure to comply with this requirement may result in damage to the device and pose a significant safety risk.
- Correctly adjust the divider ratio on the device according to the probe used. An incorrectly set divider ratio leads to incorrect measurement results, which can also lead to an incorrect risk assessment. This is particularly critical when working with high voltages.
- Make sure that the cables are arranged correctly to avoid tripping hazards. Tripping can not only lead to injuries, but can also cause the appliance to fall and be damaged.

1.1.5. Beware of dangerous voltages

- Voltages of more than 30 V AC RMS, 42 V AC peak or 60 V DC are considered dangerous contact voltages and can cause life-threatening electric shocks. Ensure that only suitably trained persons with sufficient experience conduct measurements on these voltages. Inexperienced users should never carry out such work without expert supervision.

1.1.6. Maintenance

- Do not open the housing of the oscilloscope. Repairs may only be carried out by qualified service personnel. Opening the device may expose you to dangerous internal voltages.
- Only clean the appliance with a dry cloth. Do not use any aggressive cleaning agents or solvents that could damage the surface or penetrate the inside of the appliance.
- Switch off the appliance and disconnect it from the mains before replacing the fuse. Use a fuse of the type and specifications indicated on the rating plate of the appliance. An incorrect fuse can cause damage to the appliance or impair the protective function.

1.1.7. Intended use

- Only use the product in accordance with the intended applications and within the specified performance limits. Using the oscilloscope for applications outside its specifications can lead to incorrect measurement results and hazards for the user.

24211 Preetz, Handelsweg 16, Germany / Tel. +49 4342 90786-0 / service@batronix.com / www.batronix.com

1.2. Legal provisions

1.2.1. CE labelling

- This oscilloscope fulfills the requirements of the CE marking and complies with the European directives for electromagnetic compatibility (EMC) and electrical safety. This ensures that the device can be used safely in the European Union.
- The declaration of conformity is included in the scope of delivery and can be provided on request. Keep this document in a safe place, as it may be important for warranty claims and proof of compliance with legal requirements.

1.2.2. WEEE Directive

- The device is subject to the European WEEE Directive (Waste Electrical and Electronic Equipment). Dispose of the oscilloscope at the end of its service life in accordance with the local regulations for the disposal of electrical and electronic equipment. Improper disposal can cause environmental and health hazards.
- For further information on disposal, please contact your local waste disposal center or the manufacturer. Use the designated take-back systems to ensure environmentally friendly disposal.

1.2.3. RoHS Directive

- This oscilloscope complies with the RoHS Directive (Restriction of Hazardous Substances) and does not contain any banned hazardous substances in concentrations that exceed the legal limits. This reduces the environmental impact and the risks to health during disposal.

1.2.4. International standards

- The oscilloscope fulfills international standards for electrical safety and EMC, including IEC 61010 (Safety requirements for electrical equipment for measurement, control, and laboratory use) and IEC 61326 (EMC requirements for equipment for measurement, control, and laboratory use). These standards ensure that the device can be operated safely and reliably worldwide.

We recommend that you read this manual thoroughly and follow the instructions carefully to ensure safe and effective use of your oscilloscope. If you have any further questions or technical problems, please do not hesitate to contact our customer service.

Successful measurement and analysis!

2. Commissioning

In this chapter you will find all the necessary steps to correctly unpack, set up and commission your new oscilloscope. Please follow the instructions carefully to ensure safe and optimal use of your device. By following these instructions, you will ensure that your oscilloscope functions optimally and has a long service life. If you have any further questions or problems, please contact customer service.

2.1. Unpacking and checking

2.1.1. Unpacking

- Carefully open the packaging and remove the oscilloscope and accessories.
- Keep the packaging materials in case you need to transport or return the device at a later date.

2.1.2. Check

- Ensure that all components specified in the scope of delivery are present:
  - Oscilloscope
  - Mains cable
  - 4 probes with accessories
  - Hexagon socket screwdriver, 4 mm
- Check the device and all accessories for visible damage that may have occurred during transport. Report any damage or missing parts to your dealer or Batronix immediately.

2.2. Installation site

2.2.1. Choice of location

- Select a dry, clean, and well-ventilated location for the oscilloscope.
- Avoid direct sunlight and extreme temperatures.
- Ensure that the location is free from dust, moisture, and chemical vapors.

2.2.2. Surroundings

- Place the device on a stable, level surface that can safely support the weight of the oscilloscope.
- Maintain a minimum distance of 10 cm from other appliances to ensure sufficient air circulation.

2.3. Setup

2.3.1. Positioning

- Position the oscilloscope so that the front and right-hand side are easily accessible and the controls are within easy reach.
- Make sure that the ventilation openings are not blocked to prevent overheating.

2.3.2. Accessories

- Connect the measuring probes and other accessories in accordance with the instructions in the corresponding operating instructions.
- Ensure that all connections are secure and correctly connected.

2.4. Notes on the measuring station

2.4.1. Workplace organization

- Keep the measuring station clean and tidy to avoid accidents.
- Make sure there is enough space to handle the cables, probes and accessories safely.

2.4.2. Security

- Make sure that the measuring area is free of obstacles that you could trip over.
- Make sure that no liquids can get near the oscilloscope.

2.5. Connection to the power supply

2.5.1. Mains connection

- Check that your mains voltage matches the specifications of the appliance.
- Use the mains cable supplied to connect the oscilloscope to the power supply.

2.5.2. Grounding

- Ensure that the appliance is properly grounded to avoid electric shocks.
- Only use tested and intact mains cables.

2.6. Switching on and off

2.6.1. Switching on

- Switch on the oscilloscope by setting the power switch on the back of the device to the "I" position and then pressing the power button on the front.
- Wait until the device has fully booted up.

2.6.2. Switching off

- Switch off the oscilloscope by pressing the power button on the front panel. As soon as the screen is switched off, you can move the power switch on the back to the off-position.
- Disconnect the appliance from the power supply after switching it off if it will not be used for a longer period of time.

24211 Preetz, Handelsweg 16, Germany / Tel. +49 4342 90786-0 / service@batronix.com / www.batronix.com

3. First steps

3.1. Overview of the device

3.1.1. Front view

1. Touch screen with glass surface
2. Run/Stop and Single button for recording control
3. Rotary knobs for controlling various parameters. Knobs 1 and 3 for the offset settings have a finer detent, while knobs 2 and 4 for scaling settings have a much more noticeable detent.
4. Power button for starting and shutting down the oscilloscope. Please also note the power switch on the back of the device.

3.1.2. Side view

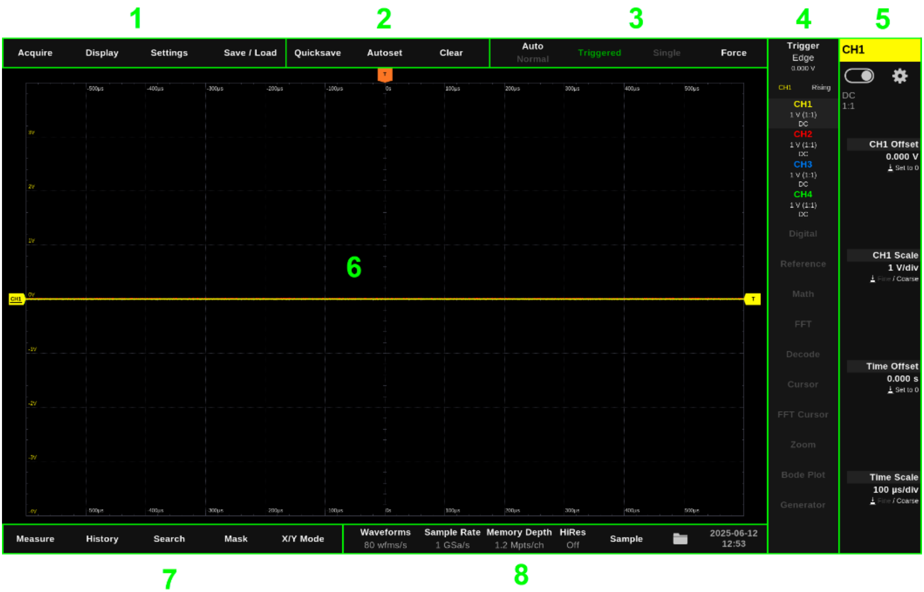
1. Connection pins for probe compensation. Please refer to the chapter on probe compensation.
2. Sockets for connecting the optional Magnova logic analyzer modules. Up to two modules with eight digital channels each can be connected.
3. BNC sockets for analog signal acquisition.
   Please note the maximum voltage load of the inputs as annotated and refer to the device data sheet for further information.
4. USB 3.0 host connection, to connect input devices such as mouse, keyboard, USB sticks, an external touch display or a touch-enabled projector. There are two further ports on the back of the device.

3.1.3. Rear view

1. Cover for retrofitting the Magnova function generator module.
   Also used as a maintenance opening for easier access to the battery for the built-in real-time clock. A screwdriver with a TX10 head is required to loosen and tighten the screws.
2. BNC sockets "AUX OUT" and "10 MHz REF IN"
   a. Pulses can be output via the AUX OUT socket when triggering or when the mask test is successful or fails. The 10 MHz reference clock used for sampling can also be output. The impedance is 50 Ω.
   b. An externally generated 10 MHz reference clock can be applied via the 10 MHz REF IN socket, e.g., to keep the sampling clocks synchronized with the reference clock of an external function generator. The signal is terminated with 50 Ω.
   Requirements: 10 MHz ± 20 ppm with 0.5 Vpp to 2.4 Vpp at 50 Ω input impedance
3. Connections for external peripherals
   a. Two additional USB 3.0 host ports, e.g., for mouse, keyboard, USB sticks, an external touch display or a touch-enabled projector.
   b. USB 3.0 device port for connection to a PC
   c. Gigabit Ethernet for connection to a network
   d. DisplayPort output with 1920 x 1080 pixels resolution (Full HD, 16:9) for connecting an external display or projector.
4. Stand and hinge with adjustable torque. It may be necessary to readjust the torque initially and with frequent adjustment. A suitable offset screwdriver for the 4 mm hexagon socket screws is included.
5. VESA mount, 75 x 75 mm, M4 thread
6. Opening for standard security lock, 3 x 7 mm (in aluminum with 2 mm wall thickness)
7. Mains socket with mains switch and fuse compartment

3.2. User interface

The user interface consists of the areas listed below. By activating further functions, additional areas can be displayed depending on the context.



1. You will find the menu bar at the top left. The contents of these menus are described in more detail in the following chapters; here is a brief overview:
    a. Acquire: Settings for the measurement run, e.g., the acquisition mode and the memory depth.
    b. Display: Settings for displaying the measurement, e.g., the intensity of the display, the color scheme, and the measurement grid.
    c. Settings: Device settings, e.g., the language setting.
    d. Save / Load: Saving and loading waveforms, screenshots and much more.
2. The shortcut buttons directly trigger a specific functionality:
    a. Quicksave: Saves one or more files at the touch of a button according to the pre-configuration in the "Save / Load" area.
    b. Autoset: The applied measurement signals are analyzed, and the vertical scaling and time scaling are preset as appropriately as possible.
    c. Clear: Deletes previously recorded waveforms as well as corresponding waveform and FFT displays.

3. In the recording control area, you can view the status of the trigger and specify trigger settings.
    a. Auto / Normal: Switching automatic triggering on and off.
    b. Triggered / Waiting / Roll-Mode / Running (Pre-Trigger): This field shows the current status of triggering. It also allows you to start or stop automatic detection with a single touch.
    c. Single: Initiates a single recording based on a mandatory corresponding trigger event.
    d. Force: Completes a recording in progress regardless of a trigger event.
4. In the control selection, channels and measurement functions can be switched on and off and can be selected for display in the settings area. In addition to this, corresponding settings windows can be accessed as will be described below.

   To activate a previously deactivated element, simply tap it. To deactivate, double tap the corresponding button or use the slider described in section 5. Activated elements are displayed in color/light, deactivated areas appear dark grey.

   After tapping an element, a gearwheel is briefly displayed next to it, which can be used to call up the corresponding settings window. A plus sign is also displayed for the Reference, Math and Decoder elements. It may be used to add another channel of the corresponding type.
5. The channel or function currently selected in the "Control selection" is displayed at the top of the settings area.

   Below this, you will see a slider for switching the channel or function on and off and you can open the corresponding settings window using the gearwheel symbol to the right of it.

   The settings for the channel or measurement function selected in the control selection are displayed directly next to the rotary controls. The values can be set using the rotary control knobs; alternatively, an additional input screen can be opened by tapping on it. The rotary controls also have context-based push button functions, which are also described and displayed.
6. The signal curves of all channels are shown in the measurement curve display area.

   The zero line markers of the channels are located on the left and an orange marker with a "T" for the trigger time is located at the top. You can tap the marker with your finger, hold it and drag it to the desired position.
7. At the bottom left you will find measurement and other functions:
    a. Measure: Various automatic measurements
    b. History: Display of individual recordings
    c. Search: Search for specific signal shapes
    d. Mask: Mask test
    e. X/Y Mode: XY representation of two channels

8. The following information is displayed in the status bar:
    a. Waveform capture rate in recordings per second (wfms/s) or time coverage in percent. You can switch between the two by tapping on them.
    b. Sampling rate: The number of measuring points written to the memory per second.
    c. Memory depth: The number of measured values saved per channel and per recording. You can open a configuration window (see 5.1) by tapping.
    d. HiRes: Arithmetic averaging of consecutive sample points can be enabled via the „Acquire“ menu or by tapping the status bar. This function can be used for noise reduction and increased signal resolution, but it acts like a low-pass filter. For details, see section 5.1 „Acquire.“
    e. Acquire Mode: The acquisition mode can be set via the "Acquire" menu or by tapping in the status bar.
    f. If a USB storage device has been detected, a USB stick symbol is displayed; otherwise a folder symbol is visible. You can open the file browser by tapping on the folder symbol. From the file browser, you can also safely disconnect USB storage devices before removing them.
    g. Date and time: Tap to open the settings window and adjust the entries if needed. If an internet connection is available, date and time can be synced with a time server.

4. Basic operation and settings for measurement curve recording

4.1. Introduction of horizontal and vertical system

In the digital oscilloscopes commonly used today, an analog-to-digital converter (ADC) converts analog signal voltages into digital quantization levels. Depending on the oscilloscope, ADC resolutions of 8, 10 and 12 bits are common, with the analog signal being converted into 256, 1024 or 4096 quantization stages. These stages are distributed over the entire input voltage range of the ADC and are displayed on the screen over the entire vertical display range.

Oscilloscopes can amplify or attenuate signal voltages in their front end (the input range up to the ADCs) to adapt different voltage levels from a few millivolts up to hundreds of volts to the input voltage range of the built-in ADC.

In the oscilloscope front end, the signal can also be assigned an offset voltage, e.g., to shift negative signal voltages into the input voltage range of the ADC. The measurement signal moves vertically on the screen when the offset voltage is set.

The horizontal axis represents the progression over time. The time scaling can also be set over a wide range. Depending on the oscilloscope, a few picoseconds up to thousands of seconds can be displayed across the width of the screen.

The display is divided into a grid of "divisions" (short form: divs). With the Magnova, there are eight divisions in the vertical and twelve divisions in the horizontal. The vertical and horizontal scaling refers to these divs, with settings typically given as 2 V/div and 1 ms/div.

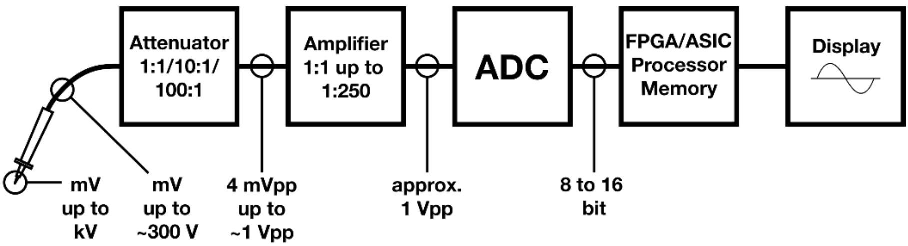


Illustration: From the probe through the oscilloscope front end to the display

4.2. Use of probes

Ensure that the probes are connected to the BNC sockets of the oscilloscope and that the correct channel settings are configured before contacting a test object with the probe! It is of the utmost importance to adhere to all safety instructions!

Apply the appropriate settings (divider factor and termination) in the "Analog Channels" settings window. To do this, first tap on one of the analog channels (CH1 to CH4) in the control selection and then on the gearwheel symbol that appears on the left. If a channel is already selected, the settings window can also be accessed by tapping the gearwheel symbol in the top right-hand corner of the screen.

4.2.1. Probe compensation

Before using the probes, they should always be compensated to match the oscilloscope input.

Without suitable compensation, the signals will be distorted and displayed incorrectly, resulting in inappropriate measurement results.

The compensation of a probe is carried out in the following steps:

- Familiarize yourself with the instructions supplied with the probe.
- Connect the probe to one of the oscilloscope's inputs.
- Connect the probe to the compensation signal that is provided on the Magnova via the probe inputs on two contact pins. Ensure that the signal and earth are connected appropriately.
- The compensation signal is a square wave signal with 3.3 Vpp and 1 kHz. You can use the „Autoset“ function for automatic adjustment or manually set the channel scaling, time scaling and trigger so that one to two periods are visible on the screen providing a stable image.
- The probes have a small adjustment screw near the tip of the probe or on the BNC connection. A small adjustment tool (slotted screwdriver) is included with the probes for adjustment.
- Turn the compensation screw until the square wave signal displayed is correct. A correctly compensated signal shows clear, rectangular edges without overshoot or undershoot.

An over-compensated signal shows pronounced peaks (overshoot) at the edges.

An under-compensated signal shows rounded corners or dips (undershoot).

4.3. Analog channel vertical settings

After tapping an analog channel in the control selection, the rotary controls are assigned the Channel Offset, Channel Scale, Time Offset and Time Scale.

You can now change these settings by turning the rotary control or by tapping on the corresponding text or value next to it to open a number editor and enter new values directly.

4.3.1. CH1/2/3/4 Offset

The offset shifts the signal up or down along the vertical axis. This allows the signal position on the screen to be adjusted so that, for example, two channels can be displayed one below the other or a channel can be shifted to a desired position despite the DC voltage component.

Please note that the offset is normally applied in the analog front end. Only by doing this can the AD resolution range be fully utilized for the screen height. However, if the offset setting is subsequently changed for a measurement that has already taken place, this measurement curve is shifted based on the data already stored in memory. If signal components were outside the AD converter range during the original measurement, these remain limited accordingly even if they are subsequently shifted.

4.3.2. CH1/2/3/4 Scale

The scaling setting influences how large or small the signal appears on the screen vertical. Small signals can be enlarged, large signals can be reduced in size.

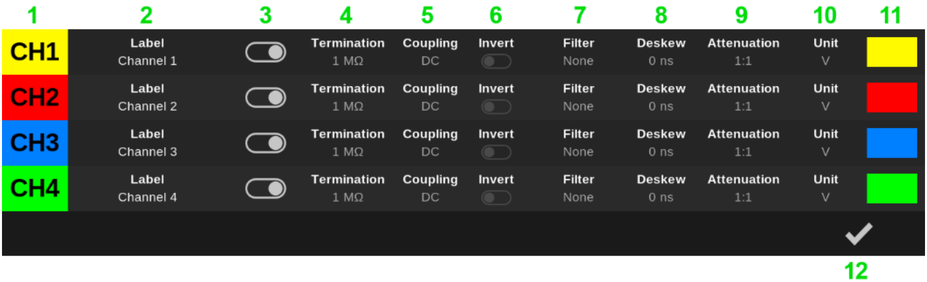
To do this, the input attenuators and input amplifiers in the analog front end are set so that the signal has the appropriate size at the AD converter input and therefore also on the screen.

To optimize the accuracy of the ADCs, you should scale signals so that they occupy a large part of the screen verticals (and therefore a large part of the input voltage range of the ADC).

Please note that scaling by signal amplification and attenuation normally takes place in the analog front end. Only in this way can the AD resolution range continue to be fully utilized for the screen height. However, if the scaling setting is subsequently changed for a measurement that has already taken place, this measurement curve is adjusted by the software. As a result, no new signal details can be created and truncated signal information remains limited accordingly.

4.4. Analog Channels settings window

In the control selection, tap on one of the analog channels (CH1 to CH4), then tap on the gearwheel symbol that appears to the left to open the Analog Channels settings window. If a channel is selected, you can also tap on the gearwheel symbol in the top right-hand corner of the screen to open the settings window.



1. Each channel has a designation (CH1, CH2, ...) which is also displayed on its marker to the left of the measurement curve display area.
2. Labelling makes it possible to assign user-defined names or designations to the channels. This function is useful for quickly and clearly identifying the signals of different channels, which can increase clarity and efficiency when analyzing signals.

   The labelling is displayed below the zero line of the channels. Note that in the default setting, the labelling only appears for a few seconds when the channels are moved. In the "Display" menu, you can alternatively set this information to be displayed permanently or not at all.
3. The channel display can be switched on and off using the slider. Please note that channels remain activated even when the display is switched off, e.g., if they are used in measurements, in the FFT or as a trigger source.
4. The inputs can be terminated with either 50 Ω or 1 MΩ.

   When using the passive proves supplied, the 1 MΩ input termination must be used. For active probes with 50 Ω output and when using a BNC cable with 50 Ω source impedance, however, the 50 Ω input termination should be used.

   By matching the input impedance of the oscilloscope to the impedance of the signal source or the transmission medium (e.g., 50 Ω coaxial cable), signal reflections are minimized. Reflections can distort the signal and impair the measurement accuracy.

Observe the maximum voltage (5 Vrms, 30 Vpk) when using the 50 Ω input impedance!

Please note that when 50 Ω input impedance and 50 Ω source impedance are activated, the voltage of the source is halved by the resulting voltage divider.

5. The inputs can detect signals together with their DC voltage component or only the AC voltage component.

   DC coupling (direct current coupling) allows both direct voltage and alternating voltage components of the signal to pass through.

   AC coupling, on the other hand, only allows AC components of the signal from approx. 7 Hz to pass by filtering the DC component and frequencies below approx. 7 Hz through a capacitor in the signal path.

   AC coupling can be used, for example, to measure the noise on a DC supply voltage. The DC voltage component is blocked, the vertical scaling can be set to a few mV/div and the noise can thus be displayed and measured as large as possible.

   Please note that with AC coupling, the divider factor of passive probes is no longer effective and the voltage is applied to the device with a factor of 1:1! Be sure to observe the input voltage limits here, as overvoltage can damage the oscilloscope!
6. The measured signals can be digitally inverted directly after the A/D conversion. Positive voltages are displayed as negative and vice versa (e.g. +1 V becomes –1 V).
7. The 200, 100, 50 and 20 MHz filters can be used to attenuate higher frequency components of the input signal. These filters are used to suppress high-frequency interference and noise that could affect the accuracy and clarity of the signal view or the trigger capability. Please note that a filter can also remove relevant signal components and distort the signal to be measured.

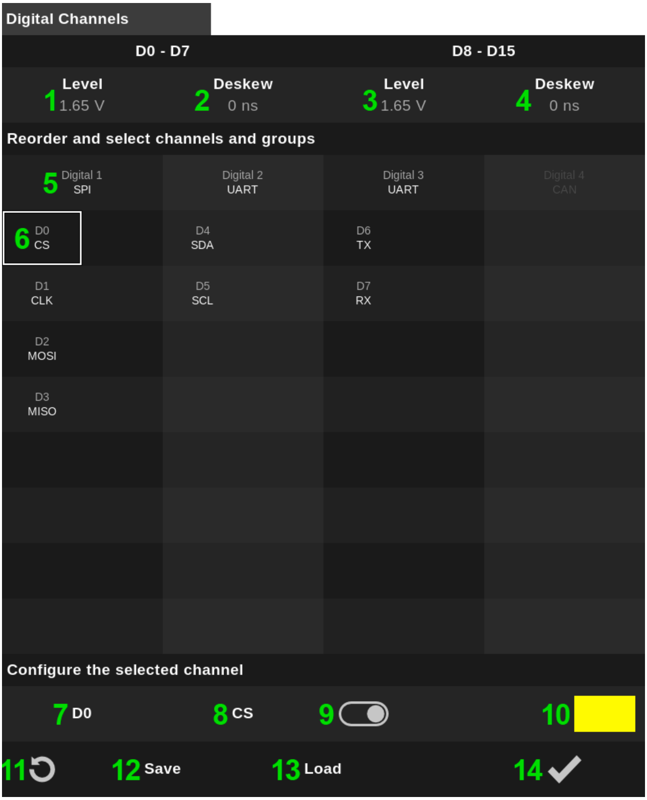
   With the amplitude filter, the natural low-pass effect of the oscilloscope can be corrected via digital filters. The frequency response is raised and corrected close to the device bandwidth. However, this can lead to a preshoot and overshoot in the signal display, particularly with steep edges (below 2 ns).
8. Channels whose input signals have different runtimes can be synchronized with each other using the runtime adjustment (deskew). This is helpful, for example, if measuring cables of different lengths or different probes are used.
9. The divider factor specifies the ratio with which the input voltage is divided or attenuated by the probe. Set the setting to suit your probe so that the measured values are displayed appropriately converted.

   Common divider factors are, e.g., 1:1, 10:1, 100:1, and others, which can be selected directly here. In the "Other" setting, any other factors from 1,000,000:1 to 1:1,000,000 can be set.
10. The display unit can be customized for voltage or current probes.
11. The channel color can be adapted according to your own wishes, e.g., to match the probe color coding rings.
12. Closes the window.

4.5. Digital channel settings window

First connect one or two of the BMO-MSO logic probes (optional accessories).

In the control selection, tap on “Digital”, then tap on the gearwheel symbol that appears to the left to open the Digital Channels settings window. If a „Digital“ is selected, you can also tap on the gearwheel symbol in the top right-hand corner of the screen to open the settings window.



1. Switching level (threshold) for D0-D7: Defines the voltage level at which the signals at the first logic probe (upper connection, D0-D7) are interpreted as high.
2. Deskew: Used for time alignment of two logic probes, e.g. with different cable lengths. In addition, an adjustment can be made with the analog channels.
3. Level for D8-D15: As under point 1, but for the second logic probe (lower connection, D8-D15).
4. Deskew for D8-D15: As under point 2, but for the second logic button head (lower connection, D8-D15).
5. Digital groups: Up to four digital groups can be set up and configured independently. After tapping a group (digital 1-4), the settings are made using buttons 8-10 (name, activation, color).

6. Digital channels: After tapping a channel (D0-D15), it can also be configured using buttons 8-10 (name, activation, color).

   The channels can be freely distributed across the four groups and can also be rearranged within a group - simply by dragging and dropping.
7. Selected element: Displays the currently selected digital group or digital channel.
8. Name: Tap to open an editor for renaming. If digital groups are named first, the system automatically suggests suitable channel names (e.g. for group name „SPI“: CLK, MOSI, MISO, CS).
9. On/off switch: Activates or deactivates the selected group or channel.
10. Color selection: Sets the color of the selected group or channel.
11. Reset: Resets all digital channel settings to the factory settings: names are deleted, channels are deactivated and moved to the first group.
12. Save settings: Saves the current configuration of the digital channels to a file.
13. Load settings: Loads a previously saved digital channel configuration from file.
14. Closes the window.

4.6. Trigger system introduction

When in "run mode", the oscilloscope continuously measures the input signal and displays it on the screen. The trigger system is used to display signals from subsequent signal sequences at the same horizontal position (the time axis). The signal thus "stands" on the screen with a fixed time reference and can be easily viewed and analyzed.

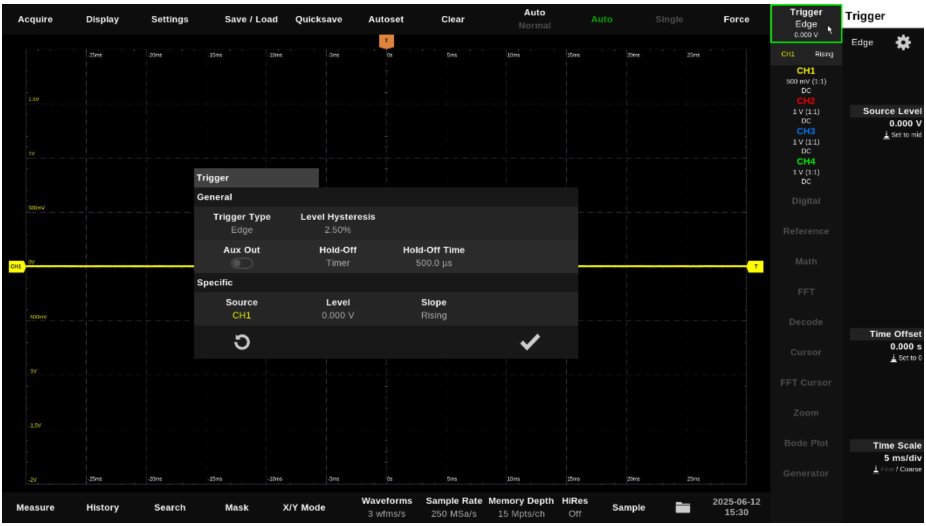
The trigger system of an oscilloscope allows a variety of settings for the trigger condition. If this trigger condition is recognized, the measured signal is displayed on the screen with the appropriate alignment on the horizontal axis. The horizontal position (at which the trigger condition was recognized) can be shifted to the left and right, even far outside the screen.

The auto-trigger option allows you to view and assess the signal before the trigger system is set correctly. If this is activated, signals are displayed several times per second even without triggering. These usually have no time reference and are positioned on the time axis without taking the signal shape into account. However, they help to estimate and find the correct trigger setting.

4.7. Trigger settings

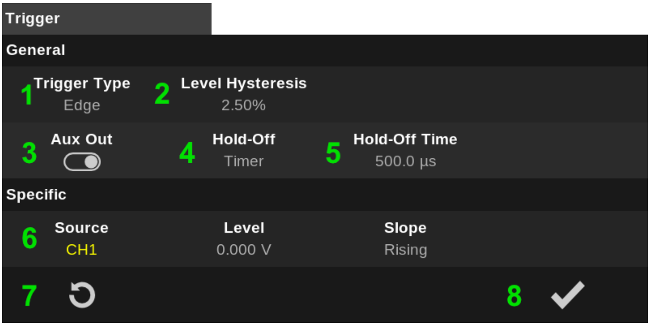
After tapping "Trigger" at the top of the control selection, the rotary controls are assigned the trigger level, a second level if necessary, the time offset and the time scale.

You can now change these settings by turning the dials or open a number editor to enter new values by tapping on a setting.



4.8. Trigger settings window

Open the trigger settings window by tapping on "Trigger" in the control selection and tapping on the gearwheel that briefly appears next to it. As long as the trigger is selected, you can also tap the gearwheel in the top right-hand corner to open the settings window.



4.8.1. Common trigger settings

1. First, select the desired trigger type. The available settings will then automatically adjust according to the selected type.
2. The level hysteresis prevents the trigger from being released unintentionally, e.g., due to noise.

   The trigger can only be triggered again if the signal level has moved far enough away from the trigger level according to the hysteresis setting before it reaches the trigger level again.

   The setting is made as a percentage of the screen verticals.
3. When the trigger Aux Out option is activated, the oscilloscope emits a positive pulse at the AUX OUT output each time it is triggered.
4. The Hold-Off setting allows you to define either the least accepted time span between one trigger event and the next or the least amount of trigger events that must have passed before the trigger is released again.
5. Depending on the Hold-Off setting, the time or the number of triggers can be set.
6. Depending on the trigger type, there are further settings such as the trigger source, level, slope, and timing specifications.
7. Sets the default settings for the selected trigger type.
8. Closes the window.

4.8.2. Edge Trigger

The edge trigger occurs when the signal of an analog channel crosses a defined threshold level in a specified direction.

In addition to analog channels, digital channels and the AC power line (AC Line) can also be selected as trigger sources. For digital channels, triggering occurs on a logic level change in the selected direction. When using the „AC Line“ as the source, triggering is synchronized with the mains frequency (e.g. 50 or 60 Hz).

Available trigger directions are „Rising“, „Falling“, „Both“, and „Alternate“. In „Alternate“ mode, triggering switches between rising and falling edges — after triggering on a rising edge, the next trigger will only occur on a falling edge, then rising again, and so on.

The edge trigger is the simplest and most frequently used trigger type. Due to its simple handling, the edge trigger is often the first step before more complex trigger types such as Nth edge, timeout or protocol trigger are used (if appropriate).

Typical application examples:

- General signal analysis: Fast acquisition of periodic or repetitive signal shapes such as square, triangular or sine waves.
- Analyzing digital signals: Triggering on rising or falling edges of a clock signal or data stream, e.g. with UART, SPI or I²C.
- Switching behavior during switching processes: Analysis of the switch-on behavior of power supply units, clock generators or logic gates where characteristic edges occur.
- Simple debugging: Initial orientation when connecting unknown signals or for quickly localizing errors in circuit behavior.

4.8.3. Nth edge Trigger

The Nth-Edge trigger occurs after a user-defined number of signal edges.

The “Idle time” setting can be used to define the minimum signal idle time, which must be before the edge counting begins. The counter for the Nth edge triggering only starts when the signal shows no edge changes for the defined time. This allows recurring signal sequences to be reliably recognized and triggered on a specific edge within a burst.

Typical application examples:

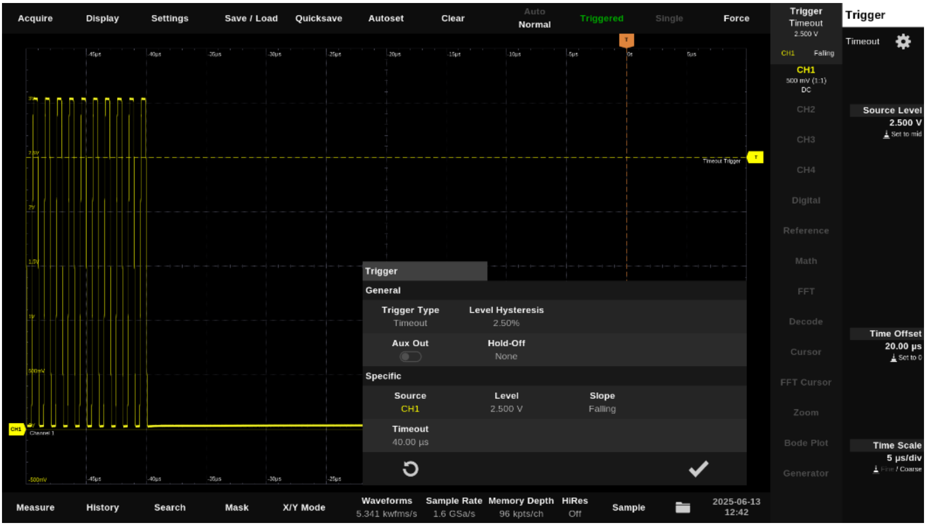
- Avoiding unstable triggering with recurring patterns: In cases such as the following screenshot, a „normal“ edge trigger would trigger on the first detected edge - which (depending on the acquire settings and pause time between bursts) may not necessarily be the first edge of the desired burst signal. Triggering on different edges of the burst signal leads to an unstable or „wandering“ display.
- Triggering on a specific edge within a burst: If a signal contains several consecutive edges in quick succession (e.g. a start burst or sync signal), it is possible to trigger specifically on the nth edge within this pattern - e.g. on the 5th rising edge.
- Bit-selective triggering for simple serial protocols: With user-defined or non-standardized serial data formats, the nth edge can be used to trigger on the 3rd bit after a start signal, for example.
- Triggering on clock edges after idle: For signals that supply clock edges again after a pause, a specific edge can be triggered after the clock resumes - e.g. to capture the first data packet after inactivity.

4.8.4. Timeout Trigger (Dropout)

The timeout trigger is triggered if the signal does not pass through the set trigger level again for a definable time after an edge. This allows signal pauses or standstills in the data stream to be reliably recognized.

Typical application examples:

- Detection of bus pauses or inactivity phases: In serial communication protocols (e.g. UART, SPI, I²C), defined pauses often occur between data packets. The timeout trigger can be used to specifically trigger the end of a data block or the start condition of an inactivity phase.
- Monitoring of watchdog functions or control pulses: In safety-critical systems, a missing pulse (e.g. from the watchdog) can be an error condition. This condition can be reliably detected with the timeout trigger.
- Analyzing signal interruptions or line failures: The trigger is suitable for detecting signal interruptions, e.g. in the event of loose plug connections or interruptions in the transmission path.
- Trigger for clock loss in clock lines: In digital circuits, the timeout trigger can be used to detect the absence of a clock signal - for example, if a clock generator fails or is stopped.



4.8.5. Delay Trigger

The delay trigger enables triggering on signal sequences in which a defined time delay between two events is to be taken into account. It works with two independent trigger sources, each of which can have its own level and direction settings (rising/falling).

As soon as the first defined event is recognized, a time measurement starts. The trigger is activated when the second event occurs in relation to the first within a defined time frame. The conditions „less than“, „greater than“, „within“ or „outside“ an adjustable time window can be selected.

The delay trigger is particularly useful if simply recognizing an edge is not sufficient and the time ratio of two signal events is also decisive.

Typical application examples:

- Timing troubleshooting: Detection of timing problems in which a subsequent event occurs too early or too late after an output signal - for example in clock data synchronization.
- Protocol analysis with time condition: Checking whether a data packet arrives after a start signal within a certain period of time - e.g. in UART or SPI communication.
- Setup and hold time violations: Analyzing digital circuits for violations of time dependencies between control and data signals.
- Status detection with defined sequence: Triggering only when two specific signal changes occur in a predefined sequence and with a specific time interval.

4.8.6. Pulse Trigger (Pulse-Width)

The pulse trigger enables triggering on pulses with a specific polarity (positive or negative) and duration. It specifically recognizes signal states in which a pulse is shorter, longer or within/outside a specific time window.

The pulse trigger is particularly useful for reliably detecting time-defined individual events that would easily be overlooked with standard edge triggers.

Typical application examples:

- Detection of glitches or spikes: triggering on very short unwanted pulses, e.g. caused by faults or errors in digital circuits.
- Monitoring pulse widths: Ensuring that control signals such as PWM pulses are within permissible time ranges.
- Detection of watchdog or reset signals: Triggering on very short or very long pulses, such as those generated by safety or reset circuits.
- Validation of protocol times: Checking whether signal pulses in serial or proprietary data protocols meet certain timing criteria.

4.8.7. Interval Trigger

The interval trigger enables triggering on the time interval between two positive or two negative edges of a signal. In contrast to the pulse trigger, which measures the duration of a single pulse (either high or low time), the interval trigger records a complete signal period.

As soon as the first defined edge is recognized, a time measurement begins. The trigger occurs when the second edge occurs before, after or within/outside a configurable time window.

Typical application examples:

- Clock frequency monitoring: Triggering on unexpected frequency deviations, e.g. if the clock becomes too fast or too slow.
- Troubleshooting for signal interruptions: Detection of excessively long intervals between two events, e.g. communication interruptions.
- Detection of jitter or timing anomalies: Detection of signal instabilities where the interval between edges fluctuates.
- Analyzing pulse frequencies or pulse sequences: Validation of whether pulses occur at regular intervals - useful for encoder signals, pulse counters or trigger sequences.

4.8.8. Window Trigger

The window trigger is triggered when a signal either enters or leaves a defined voltage window. The voltage window is defined by two threshold values (upper and lower limit).

In addition, an optional time condition for staying in or outside the window can be specified. The conditions „less than“, „greater than“, „within“ or „outside“ an adjustable time window can be selected.

The window trigger is particularly useful when signals not only need to reach certain levels, but also need to move within or outside a voltage range in a time-controlled manner.

Typical application examples:

- Overvoltage or undervoltage detection: Triggering when a signal leaves a permissible range, e.g. for monitoring supply voltages or sensor values.
- Detecting voltage dips or peaks: Triggering when a critical voltage range is briefly entered, e.g. during load changes or faults.
- Validation of signal curves: Ensuring that a signal remains stable within a permitted range for a defined period of time.
- Glitch detection: Detection of short voltage peaks or dips that only briefly cross the voltage window.

4.8.9. Runt Trigger

The runt trigger recognizes so-called dwarf pulses - i.e. signal edges that do not run completely through two defined voltage levels. The signal stops between the upper and lower thresholds or returns without reaching the second voltage level.

An optional time condition can also be specified. The conditions „less than“, „greater than“, „within“ or „outside“ an adjustable time window can be selected.

The runt trigger is particularly useful when errors are caused by unstable or disturbed signal states.

Typical application examples:

- Fault analysis in digital circuits: Detection of incomplete logic levels, e.g. due to crosstalk, runtime differences or faulty drivers.
- Analyzing glitches: Detecting brief faults that cannot be evaluated as a complete high or low state, but can still cause logic errors.
- Timing errors in bus systems: Investigating whether signals are transmitted correctly and completely or whether runt pulses occur, for example in the case of transition errors.
- Validation of signal quality: Checking whether defined signal levels are reliably achieved in a digital system.

4.8.10. Slope Trigger (Slew Rate Trigger, Rise Time Trigger)

The slope trigger enables triggering based on the slope (rate of rise) of a signal. For this purpose, the time required for the signal to rise (rising edge) or fall (falling edge) from a lower to an upper threshold value is measured.

As soon as the first defined edge is detected, a time measurement begins. The trigger is activated when the second edge occurs before, after or within/outside a configurable time window.

Typical application examples:

- Detection of slow or fast signal edges: Ideal for analyzing signal integrity and switching speed in digital or analog circuits.
- Detection of overshoots or debounce problems: Triggering on unintentionally slow edges, e.g. key debounce or weak pull-up resistors.
- Validation of driver stages: Checking whether output signals comply with expected rise or fall times.
- Measurement of capacitive load delays: If signal edges are delayed by large loads, the slope trigger can respond specifically to such cases.

4.8.11. Setup & Hold Trigger

The setup & hold trigger enables the temporal analysis of two signal edges in relation to each other - typically clock and data signals. It checks whether a data signal remains stable for a certain time before (setup time) and after (hold time) a clock edge.

The setup time is the minimum time that the data signal must be stable before the clock edge.

The hold time is the minimum time that the data signal must remain stable after the clock edge.

The trigger is activated if the timing between the data and clock edge does not correspond to the specified setup and hold conditions - i.e. in the event of timing violations.

Typical application examples:

- Debugging of serial interfaces: Detection of unstable or improperly clocked data lines.
- Timing analysis for clock domain crossing: Detection of meta stabilities and errors at the transition between different clock domains.
- Verification of semiconductor designs: Classic application in chip development for checking timing criteria in logic design.

4.8.12. Pattern Trigger

The pattern trigger allows triggering based on specific signal states or combinations of multiple analog and/or digital channels.

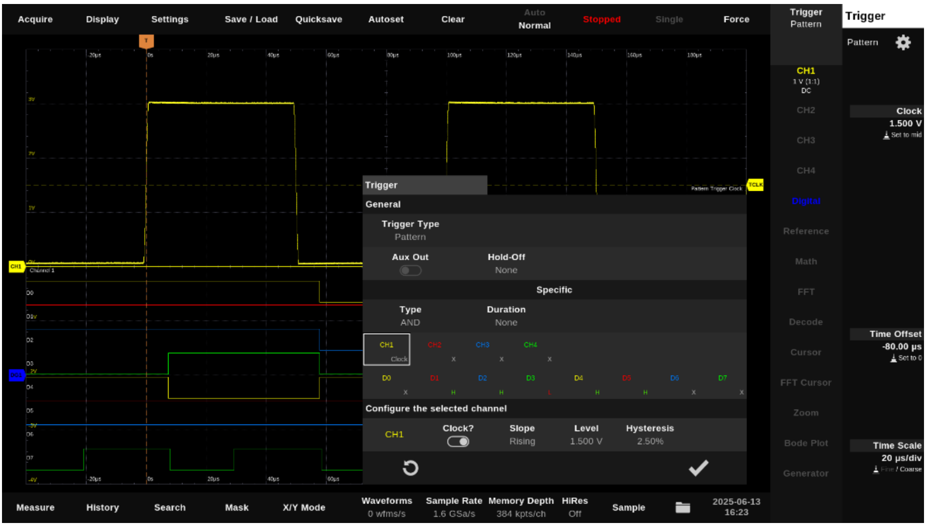
The available logic types are AND (all conditions must be met simultaneously), OR (at least one condition must be met), as well as the inverse types NAND and NOR.

Optionally, a minimum or maximum duration can be defined for which the pattern must persist in order to trigger. The available time conditions are less than, greater than, within, or outside a configurable time window.

Alternatively, one of the analog or digital channels can be used as a clock source. In this case, the logic states are evaluated synchronously with the edge of the clock signal.

Typical application examples:

- Parallel bus analysis: Triggering on a specific bit pattern on several data lines (e.g. address or data bus in microcontroller systems).
- Status detection in digital control systems: Triggering when several signals assume a defined state at the same time (e.g. safety or enable signals).
- Protocol debugging: Triggering on rare or faulty combinations in mixed signal forms (e.g. analog signal plus digital control signals).
- Synchronized triggering: Triggering on a defined pattern synchronous to an external clock source - e.g. for synchronous bus systems or clocked control systems.



4.8.13. Decoder Trigger

The decoder trigger enables triggering on serial or parallel interfaces. The signal sequences on the corresponding lines are analyzed, decoded and broken down into their protocol components, which then serve as trigger sources. Trigger conditions can be set specifically for protocol elements such as addresses, data, control bits or error events. This allows complex communication processes to be analyzed efficiently, conspicuous transmissions to be recorded in a targeted manner and recurring problems to be identified reliably.

Note: Before a trigger can be applied to a decoder, the corresponding decoder must be configured (see chapter „Decoders“).

4.8.13.1. SPI

Lane: Triggering can take place on either the MOSI or the MISO data line.

Alignment: The Word setting triggers on complete data words. The word length corresponds to the number of bits defined in the decoder setup. With the Bit setting, a number of data bits can be triggered.

Position: The „Offset“ setting can be used to define a fixed position at which the searched data must appear for triggering. The „Anywhere“ setting allows triggering as soon as the data appears anywhere in the data stream.

Length: Up to 8 data words with a total of up to 128 bits can be defined as trigger conditions.

Data words / data bits: Enter the data to be searched for in hexadecimal, decimal, binary or ASCII format. The wildcard character X allows flexible address patterns in hexadecimal and binary format, e.g. 4X for 0x40-0x4F or XX0XXX1 in binary format.

4.8.13.2. I²C

Event type: Triggering can take place on individual I²C events, addresses, data or combinations of address and data.

Supported event types are: Start, Stop, Restart, ACK, NACK, Address, Data, Address & Data

The following options are available when selecting the „Address“ or „Address & Data“ event type:

- Length: Choice of Auto, 7-bit or 10-bit.
- Direction: Read, write or read/write.
- Address: Input in hexadecimal, decimal, binary or ASCII format. The wildcard character X allows flexible address patterns in hexadecimal and binary format, e.g. 4X for 0x40-0x4F or XX0XXX1 in binary format.

The following settings can be made when selecting the „Data“ or „Address & Data“ event type:

- Position: Choice of Offset (fixed byte position) or Anywhere (any position).
- Offset: Definition of the byte position in the data stream.
- Length: Up to 8 data bytes can be defined.
- Data words/data bits: Input of the data to be searched for in hexadecimal, decimal, binary or ASCII format. In hexadecimal and binary format, X is also permitted as a placeholder, e.g. 3X for 0x30-0x3F or 1011XX01 in binary format.

4.8.13.3. UART

Triggering can take place on UART events and data.

Supported events types are: Frame Start, Incomplete Frame Start, Failed Parity Check, Incomplete Frame Completion and Data.

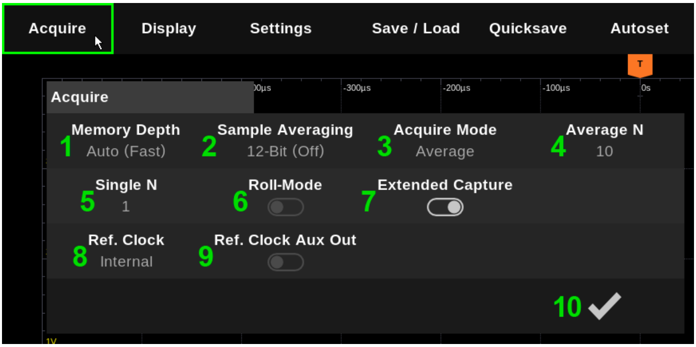
The following settings can be made when selecting the „Data“ event type:

- Position: Choice of Offset (fixed byte position) or Anywhere (any position).
- Offset: Definition of the byte position in the data stream.
- Length: Up to 8 data bytes can be defined.
- Parity: Ignore, match, mismatch
- Data words/data bits: Input of the data to be searched for in hexadecimal, decimal, binary or ASCII format. The wildcard character X allows flexible data patterns in hexadecimal and binary format, e.g. 4X for 0x40-0x4F or XX0XXX1 in binary format.

5. Menu bar

5.1. Acquire

The settings that are already used when the signals are recorded are made in the "Acquire" window. To display the window, please tap on "Acquire" in the menu bar.



1. The Memory Depth setting determines how many measuring points are saved per recording. The AD converters work continuously at the maximum speed of 1.0 or 1.6 billion samples per second. If the memory depth is not sufficient to record all samples in the configured recording time, a reduced sampling rate is automatically saved. This sampling rate used for saving the measured values is displayed in the status bar.

   The memory depth per recording can be set automatically by the software or manually using a selection list.

   a. Auto (Fast): The memory depth is set by the software so that it does not lead to a significant slowdown in the display even with slow time scaling.

   b. Auto (Max): The memory depth is set by the software so that the complete sampling rate of the AD converters is saved if possible. If the total available measured value memory is not sufficient for this, the sampling rate is also reduced in this mode before saving.

   c. Fixed memory depths: To specify a certain maximum memory depth, this is set manually using a selection list. The sampling rate used for storage is automatically adjusted by the device to the maximum possible value. Depending on the resulting sampling rate and time scaling, the "surplus" memory depth is used to store measuring points to the left and right of the display width (12 divs).

2. With Sample Averaging (HiRes), arithmetic averaging can be applied to consecutive measured values. This leads to noise suppression while at the same time utilizing an increased measured value resolution. The resulting increased effective signal resolution is also available for further acquisition and analysis functions such as triggering, measured value display and automatic measurements.

   With the Magnova, sample averaging takes place very early in the processing chain and is therefore always based on the unrestricted ADC sampling rate (1.0 GSa/s or 1.6 GSa/s), even if a reduced sampling rate is saved due to memory limitation or large time scaling.

Compared to the averaging of consecutive measurement curves (Average) described below, sample averaging has the advantage that a single signal curve can also be filtered and does not necessarily have to be triggered. Please note, however, that sample averaging acts as a low-pass filter and therefore also suppresses any desired signal components if these are within the range of the filter effect.

Resolution	Filter length	Filter bandwidth at 1.0 GSa/s sampling rate	Filter bandwidth at 1.6 GSa/s sampling rate
12.5 bit	2	~ 220 MHz *	~ 352 MHz *
13.0 bit	4	~ 110 MHz *	~ 176 MHz *
13.5 bit	8	~ 55 MHz	~ 88 MHz
14.0 bit	16	~ 27.5 MHz	~ 44 MHz
14.5 bit	32	~ 13.8 MHz	~ 22 MHz
15.0 bit	64	~ 6.88 MHz	~ 11 MHz
15.5 bit	128	~ 3.44 MHz	~ 5.5 MHz
16.0 bit	256	~ 1.72 MHz	~ 2.75 MHz

* If the model bandwidth is smaller, this is applied first.

3. The following Acquire Modes are available:
   a. Sample: With this standard setting, the measuring points determined by the ADC are used.
      If the sampling rate has to be reduced before saving, e.g. because a wide time range is to be recorded and the memory depth is limited, the sampled values are saved at regular intervals.
   b. Peak Detect: If the maximum sampling rate can be saved, Peak Detect mode behaves in the same way as Sample mode. As soon as the sampling rate has to be reduced before storage, e.g. because a particularly large time range is to be recorded and the storage depth is limited at the same time, the maximum and minimum values are stored alternately. With this setting, all peak values are retained and therefore visible.
   c. Average: With average value acquisition, an adjustable number of several measurement curves acquired in succession are superimposed and a common average value is formed for each position. This type of acquisition can lead to strong noise suppression and more precise measurement results.
      Compared to sample averaging, this averaging method also has the advantage that it does not reduce the bandwidth. Please note, however, that depending on the behavior to be investigated, a recurring signal curve and suitable triggering may be necessary.
4. If the "Average" mode is selected, the number of measurement curves to be superimposed is specified here as well.

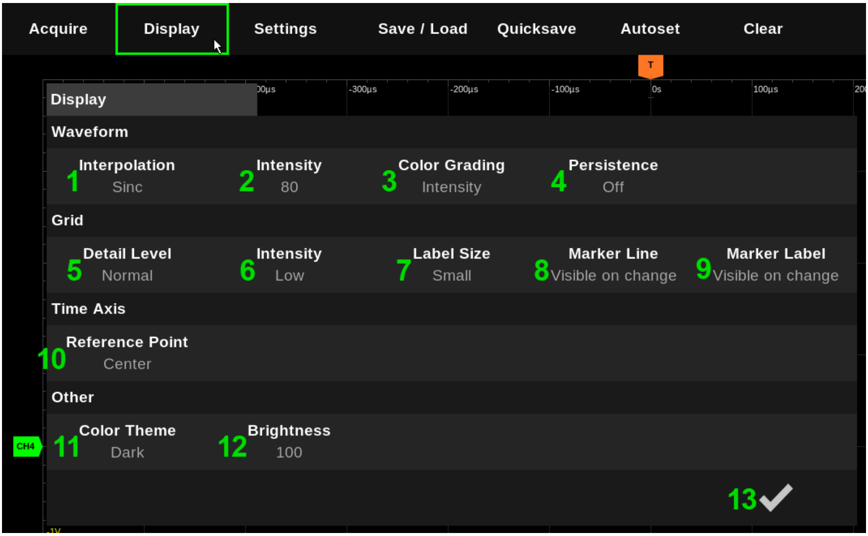
5. The single-shot function can be used to trigger a defined number of single shots (Single N).
6. „Roll-Mode“ allows the continuous display of signal curves with a slow time base. The mode can be used from a time scaling setting of approx. 50 ms/div and above. If the time scale setting is smaller, the normal display is used.

   In roll-mode, the signal runs into the measurement curve display area at the right-hand edge.
7. If "Extended Capture" is activated, a further (at least) six divs are recorded and saved to the left and right of the display area. This allows you to scroll into the areas to the left and right of the normal screen display width even after a recording has been made.
8. In addition to the internal reference clock, an externally applied reference clock can also be used as a 10 MHz reference clock to sample the signals. This allows the Magnova to run clock-synchronized to an external signal source (e.g., a function generator).

   If the optional generator module is installed, this also uses the selected reference clock.
9. The reference clock used can be provided via the AUX OUT output, regardless of whether it was generated internally or applied externally. This allows an external device (e.g., a waveform generator or another oscilloscope) to be operated clock-synchronized with the Magnova.
10. Close the window.

5.2. Display

In the "Display" window, the measurement curve display, grid, brightness, and other display properties can be customized.



1. Interpolation is used to calculate and display additional points between the values recorded by the AD converter as required.

   Example: At a sampling rate of 1 GSa/s, an AD converter measurement takes place every nanosecond. With the time scaling of 1 ns/div, only one measured value is available per div. Depending on the signal and the number of available recordings in the memory, no uninterrupted waveform becomes observable. Various options are therefore available on the Magnova to connect existing measuring points or to reconstruct the signal shape optimally:

   a. None (None): With the "None" setting, only the values recorded by the ADC are used, without additional interpolation. With fine time scaling, measurement curves without reconstructed intermediate values can be unclear or difficult to recognize.

   b. Sample and hold: With "sample and hold" interpolation, the value of each stored data point is retained until the next stored point. This leads to a possibly discontinuous / "edgy" representation of the signal between the sample points.

   c. Linear: Linear interpolation connects the stored data points using straight lines. This creates a complete representation of the signal between the actual data points, which can also deviate significantly from the actual signal depending on its characteristics.

   d. Sinc: Sinc interpolation uses a sinc filter to generate a realistic reconstruction of the actual signal between the sampling points. Sinc interpolation alone calculates up to 511 additional intermediate values in the Magnova and thus achieves a data rate of 512 GSa/s (at 1.6 GSa/s corresponding to 819.2 GSa/s).

Regarding the display, interpolation only takes place for time scales for which there is less than one measuring point per screen column.

2. The Intensity setting can be used to adjust the luminosity of measurement curves within the waveform display area. By tapping on this entry you may optionally assign the corresponding value by using the rotary knob.

   Depending on the setting and measurement signal, many waveforms can be displayed on top of each other, or many measurement points can be recorded per display column. Pixels with more frequent "hits" are displayed more brightly in the "Intensity" Color Grading setting than pixels with fewer hits.

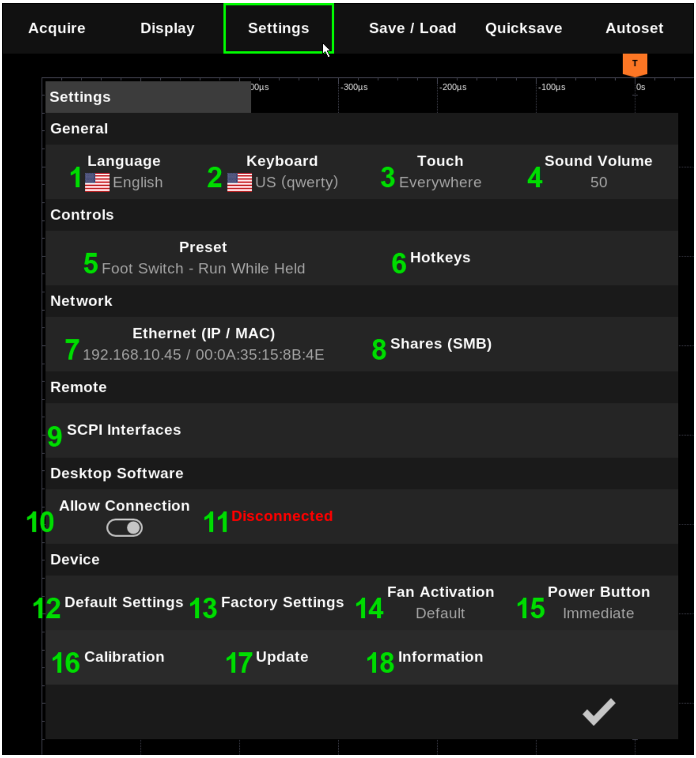
3. The following color grading options are available:
   a. Intensity: Pixels with more frequent "hits" are displayed brighter on a dark background (according to the selected color scheme) than pixels with fewer hits.
   b. Inverted intensity: Pixels with more frequent "hits" are displayed darker on a dark background (according to the selected color scheme) than the pixels with fewer hits. This setting is particularly suitable for better emphasizing rare events.
   c. 5-color heat map: The display ranges from red (many hits) via yellow, green, and cyan down to blue (few hits). The specified channel color is ignored.
   d. 7-color heat map: The display ranges from white (many hits) to red, yellow, green, and cyan, blue down to dark blue (few hits). The set channel color is ignored.
   e. 2-color gradient: The display changes from red (many hits) to blue (few hits). The set channel color is ignored.

4. Using the „Persistence“ function, recorded waveforms can be displayed on the screen for an adjustable period of time and slowly fade out during this time.

   This functionality can be useful when troubleshooting and diagnosing signal problems, for example. It makes it possible to visualize rare faults or anomalies in the signal curve for longer periods of time.

5. The Grid Detail Level can be used to set the type and number of grid subdivisions.
6. The Grid Intensity can be set to Low or High.
7. The font size on the grid can be set to small, medium or large.
8. The marker lines (trigger level, trigger position, zero position of the channels) can be displayed either permanently or only when their position is changed.
9. The "Marker Label" setting determines when the labelling of the channel zero lines shall be visible. They can be displayed permanently, only when the position is adjusted or not at all.
10. The reference point defines the point in time around which the measurement curve display is scaled when the time scale is altered.
11. The color scheme can be set to dark or light.
12. The display brightness can be set from 0 (very low brightness) to 100 (maximum brightness).
13. Close the window.

5.3. Settings



1. User interface language setting.
2. Choose between the QWERTZ and QWERTY layout of the on-screen keyboard.
3. The touch function can be restricted to the area outside the measurement curve display. This allows you to point your finger at content there without causing the display to change.
4. The volume setting for sound outputs (e.g. for the mask test).
5. Setting the function of the foot switch (optional accessory BMO-FS).
6. Keyboard and mouse buttons can be individually assigned functions in the hotkey editor.
7. Tap the Ethernet network information to open another window for the DHCP and DNS settings.
8. Network drives can be integrated here via SMB. Further information can be found in the following chapter „Setting up shares (SMB)“.
9. Display window for the Rest API, SCPI Raw, SCPI HiSLIP and USBTMC addresses.
10. The slider can be used to allow or prevent a remote connection.
11. Status display for the remote connection (connected or disconnected).

12. „Default Settings“ resets the oscilloscope to a defined default state. All basic settings such as time base, trigger, channel states, display mode and measurement parameters are reset to sensible initial values - ideal for a quick, clean restart without being affected by previous configurations. User settings such as language, volume, hotkey assignments, network drives, fan behavior and other settings are retained.
13. „Factory Settings“ resets all settings to the original factory settings. In addition to all settings, user settings such as language, volume, hotkey assignments, network drives, fan behavior and other settings are also deleted. This function is useful for troubleshooting or preparing a resale.
14. The Magnova is designed so that it can be operated without a fan at typical room temperatures. However, if required, the fan control's response threshold can be lowered so that the fan starts earlier - or alternatively set so that it runs continuously.
15. When pressing the power button, the behavior can be configured so that the device either shuts down immediately (default) or is only switched off after confirmation by tapping the button a second time.
16. The analog inputs of the Magnova can be recalibrated automatically.

    Only carry out the calibration once the oscilloscope has been in operation for at least 40 minutes and is therefore fully warmed up.

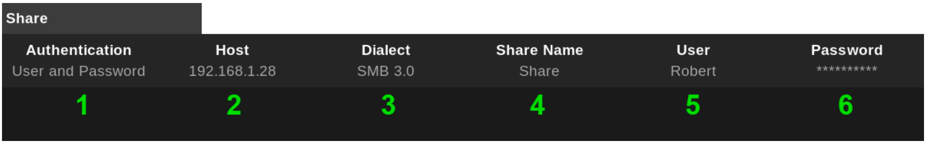
    The first step is to calibrate the offsets. This is carried out with open inputs; please remove all measuring cables or probes from the inputs.

    In a further optional step, the amplitudes can then be calibrated. This is not necessary on the user side, but we have nevertheless decided to make this functionality available to you. You will need a generator and a multimeter that can measure an AC voltage of 10 kHz with an accuracy of at least +-0.1%. You will also need the appropriate cables and T-pieces to connect the signal from the generator to all four inputs of the Magnova and to the multimeter.
17. The Magnova software can be kept up to date by using the update functionality. You can find the latest firmware version at https://www.batronix.com/magnova/en/downloads or on the Magnova product pages.
18. The information dialog shows software, firmware and hardware versions as well as the serial number of the device. In addition, bandwidth licenses can be activated and support logs saved here.

5.3.1. Setting up shares (SMB)

SMB shares enable access to network folders directly from Magnova.

The oscilloscope can access shared folders in the local network via the SMB protocol (Server Message Block), for example to save measurement data directly or to load configuration files. The prerequisite is that the target folder in the network is shared and that the user name and password are known.



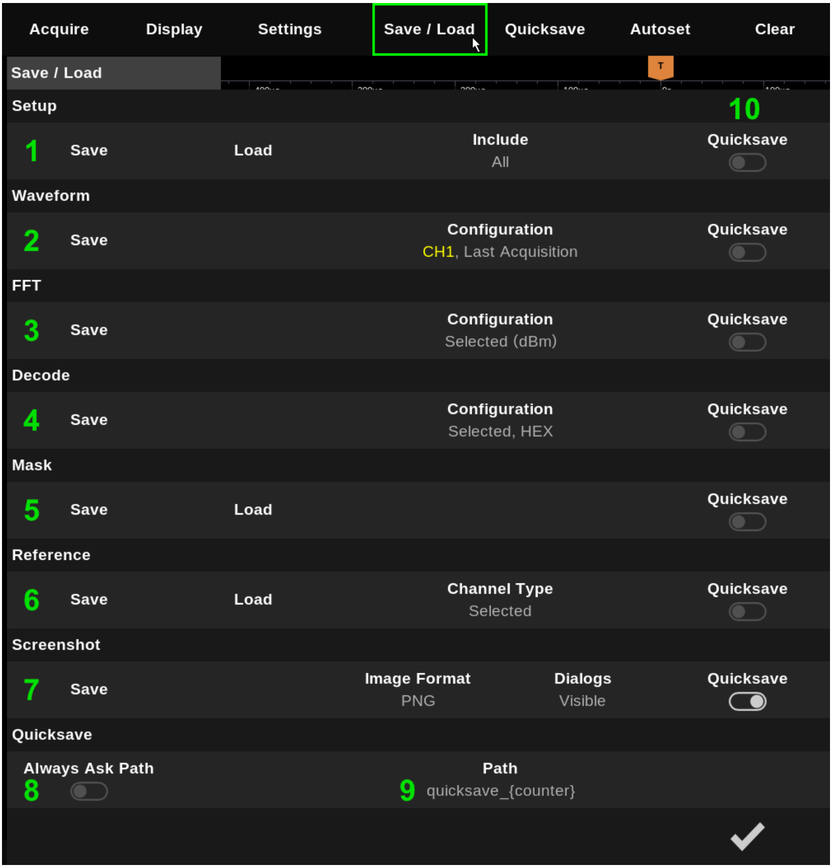
1. Authentication: Select whether the connection should be made as a guest (without user name and password) or with user name and password.
2. Host: Enter the IP address of the computer on which the share was set up.
3. Dialect: Select the appropriate SMB version according to the target system:
   a. SMB 1.0: e.g. Windows NT 4.0, XP, older NAS systems
   b. SMB 2.0: e.g. Windows Vista / Server 2008
   c. SMB 2.1: e.g. Windows 7 / Server 2008 R2
   d. SMB 3.0: e.g. Windows 8 / Server 2012 and newer
4. Share name: Enter the name of the network share (e.g. Share).
5. User: Enter the name of a user who has access to the share
6. Password: Enter the corresponding password.

Share a folder under Windows for Magnova:

1. Open the file explorer.
2. Create a new folder if required, e.g. Share in the Documents directory in the example above.
3. Right-click on the desired folder and select „Properties“
4. Switch to the „Sharing“ tab and click on „Advanced sharing...“.
5. Activate the „Share this folder“ option
6. Specify a share name (e.g. Share).
7. Click on „Permissions“ and configure access as required.

5.4. Save / Load

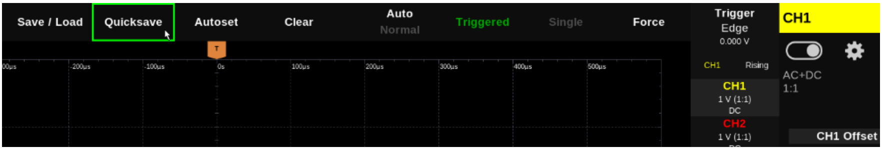
In this window, the device settings and various types of data can be saved and loaded and the behavior of the Quicksave button can be configured.



1. Setup: The current device settings can be saved and reloaded here. It is also possible to save or load only a subset of the device settings. This is useful, for example, if you only want to save or load the decoder settings but want to retain the other settings.
2. Waveform: The measurement data can be saved as a csv file in table format or as a binary file with an additional metadata csv file. The channels to be saved can be selected and the data of one, several or all recordings can be saved. All measurement points or a reduced set can be saved. The values can be saved in simple, scientific, or technical notation.
3. FFT: The FFT spectrum can be saved as a csv file in table format. The scaling can be adopted from the FFT display or specified otherwise. The values can be saved in simple, scientific, or technical notation.

4. The decoded data can be saved as a csv file in table format. The timestamp (relative to the trigger time), duration, transfer number, type, data, and error information are saved. The data can be saved in hexadecimal (HEX), decimal (DEC) or binary format (BIN) or as ASCII characters.
5. The masks created for the pass/fail test can be saved and reloaded.
6. Captured references can be saved and reloaded. You can choose between saving the currently selected references, the visible references, and all references.
7. Screenshots can be saved in PNG and BMP formats. Dialogs can be automatically hidden when saving a screenshot using the quicksave function or included in the screenshot.
8. The quicksave function can query the path and file name each time it is called or use the value preset here.
9. You can set the path and the first part of the file name for the quicksave function here. During quicksave, the files are then saved with a second part of the file name (e.g., _decode for the decoder data or _mask for the mask data) and the appropriate file extension.
10. You can use the quicksave sliders to select which data should be saved when the quicksave function is executed.

6. Further functions and recording control



6.1. Quicksave

Tapping the Quicksave button saves the files previously selected under "Save / Load".

6.2. Automatic setting (Autoset)

The Autoset function searches for existing signals on all channels and makes the best possible settings for the display.

For reliable detection, the signals should have a voltage level of more than 20 mVpp and a frequency of more than 40 Hz. Otherwise, they may not be recognized as a valid signal so that the corresponding channels are or remain switched off.

6.3. Clear

The Clear button deletes recordings and measurement data that have already been recorded.

6.4. Auto / Normal

In auto-trigger mode, signals are displayed several times per second even without triggering. These usually have no time reference and appear at different horizontal positions. However, they help to estimate and find the correct trigger setting.

In normal trigger mode, triggering only takes place when the trigger condition has been fulfilled.

6.5. Trigger status

The trigger status is displayed here. Furthermore, the recording can be started or stopped with a tap in the same way as with the "Run / Stop" button.

- Triggered: The trigger condition has been recognized and triggering has been carried out.
- Stopped: The current measurement was stopped, or a single triggering was carried out.
- Waiting: The oscilloscope waits for the trigger condition to be fulfilled.
- Auto: The trigger condition has not been met, but automatic triggering takes place using the auto trigger setting.

6.6. Single

The single button can be used to start a single recording. Recording starts as soon as the trigger condition is recognized or "Force" is tapped.

In the „Acquire“ menu, „Single N“ can be used to define a number of individual recordings that are recorded one after the other each time the single function is started.

6.7. Force

The Force button can be used to trigger an immediate recording in the "Waiting" trigger status, regardless of whether the trigger condition is fulfilled.

24211 Preetz, Handelsweg 16, Germany / Tel. +49 4342 90786-0 / service@batronix.com / www.batronix.com

7. Measurement, analysis, and display functions

7.1. Measure

Various automatic measurements can be switched on in the "Measure" window. The display of the measured values, the display of measured value curves, the measuring range and averaging can also be set here.

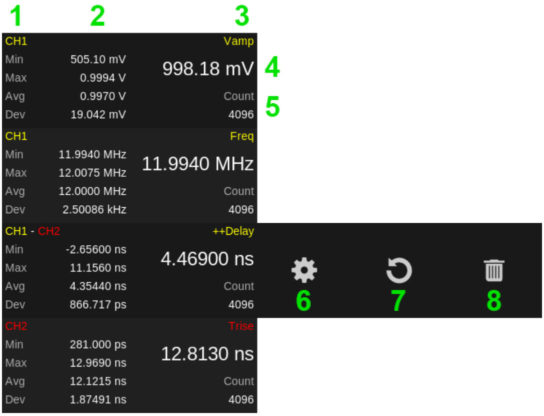
1. The display of the measured values can be switched on and off using the slider.
2. The display can be shown in compact form above the measurement curve display area or in detail, including statics, on the left-hand side of the screen.
3. The measurement results can be displayed directly on the measurement curves in the signal area.
4. The trend chart display on the oscilloscope makes it possible to visualize measurement data over longer periods of time, revealing changes and patterns in signals that might not be visible in short snapshots. This feature is particularly useful for monitoring and analyzing slow drifts, periodic fluctuations, or rare anomalies. By continuously recording, the trend chart display facilitates troubleshooting and provides an efficient way to observe the behavior of circuits over time.

5. The measurement window setting can be used to switch the area used for automatic measurements between the full screen width and a specified area.

   If the measurement window is defined based on a selection, two selection pointers appear above the measurement curve display area, which can be positioned to set the area used for the measurements.
6. In the detailed display on the left-hand side of the screen, an average value (Avg) is displayed, among other values related to each automatic measurement. The average value can be calculated either using a fixed number of measured values (Fixed) or using running average (Running), in which each new measured value is included in the average value with a suitable factor.
7. If a fixed number of measured values is desired, the average value can be calculated over 1 to 4096 measured values.
8. First select a channel here before selecting the reference level and the parameters to be measured.
9. Some parameters are calculated either based on the amplitude (Vamp), the peak-to-peak voltage (Vpp) or manually set levels.
10. The percentage value at which the lower level (Vlower) is to be determined and at which, for example, the rise time measurement begins.
11. The average level at which the signal edges are recognized.
12. The percentage value at which the upper level (Vupper) is to be determined and at which, for example, the rise time measurement ends.
13. In the "Vertical" measurement selection area, you will find voltage-based measurement parameters such as the Vamp and Vpp measurement and, to the right of this, the measured values on which these two are based (Vamp is based on Vtop and Vbase, while Vpp is based on Vmax and Vmin).
14. In the "Horizontal" measurement selection area, you will find time-based measurement parameters such as the frequency and period duration as well as counter-based measurement parameters such as the edge or pulse counter.
15. Tapping the double trash bin deactivates all selected measurements again.
16. Tapping the single trash bin deactivates all selected measurement for the selected channel.
17. Resets the measurement statistics and the trend charts of all measurement parameters.
18. Saves all settings of the window in a file.
19. Loads all window settings from a file.
20. Close the window.

7.1.1. Detailed display of measured values

If you select the detailed display in the "Measurement" window, the measured values and statistics are displayed on the left-hand side of the screen.



The display sequence of the measured value boxes can be altered by touching them or using the mouse and dragging them vertically to other positions.

The three control icons (6 to 8) are displayed for a few seconds after tapping a measured value box.

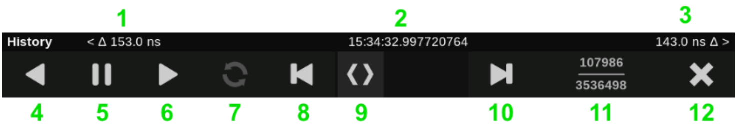
1. The channel on which this measurement is based is displayed in the top left-hand corner of each measurement box. For measurements between two channels, such as the delay measurement, both channels are displayed.
2. The minimum value (Min), maximum value (Max), average value (Avg) and the standard deviation (Dev) are displayed.
3. The measurement parameter is displayed at the top right.
4. The current measured value is displayed in a larger font.
5. The counter shows how many individual measured values have already been included in the statistics.
6. Tapping the gearwheel opens the "Measurement" window, now based on the current context.
7. The statistics and trend charts for this measured value can be reset by selecting the circular arrow. In the "Measurement" window, you will find the same symbol at the bottom left, which can reset the statistics for all measured values.
8. The measurement parameter can be removed by tapping the trash bin.

7.2. History

The Magnova saves consecutive recordings in separate memory segments ("segmented memory"). This function allows the individual recordings to be displayed and analyzed.

Tap on "History" at the bottom of the screen to activate the history display.

7.2.1. History control bar



1. The time interval to the previous recording (from trigger time to trigger time).
2. The trigger time of the displayed recording.
3. The time interval to the subsequent recording (from trigger time to trigger time).
4. Starts playback of the recordings in reverse. The playback speed can be set in five stages with a further tap.
5. Stops the playback run.
6. Starts playback of the recordings in forward mode. The playback speed can be set in five stages with a further tap.
7. If this function is activated, playback can be continued with the first recording when the last recording is reached.
8. Display the previous recording.
9. The slider can be used to navigate manually between the recordings.
10. Show the following recording.

11. Shows the number of the currently displayed record (top) and of stored records (bottom). Tapping the element opens an editor for specification of the desired record number.
12. Exits the history mode.

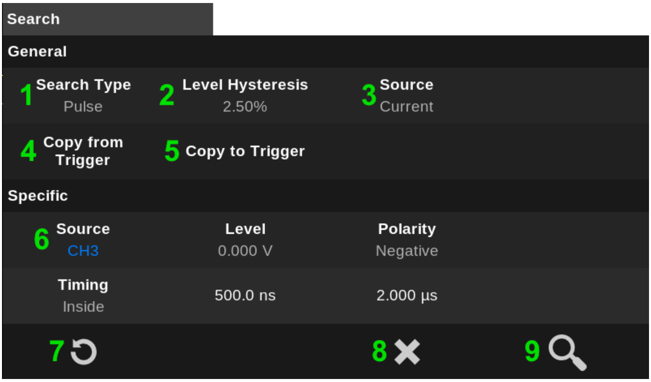
7.3. Search

You can use the Search function to search for and display specific signal sequences. The search types correspond to the trigger types, but unlike trigger functionality, search event detection can take place after the record segments have been acquired and may show several hits in one recording.

It should be noted that, unlike the actual trigger system, the search function can only access data already stored in memory and therefore cannot consider certain time sequences prior to the respective recording.

7.3.1. Search window

Tap on "Search" at the bottom of the screen to open the search window.



1. The search types correspond to trigger types (see description in the corresponding chapter). For example, you can search for and display signal curves with specific pulse lengths, slew rates or other behavior.
2. As with the triggers, a hysteresis value can also be set for the search.

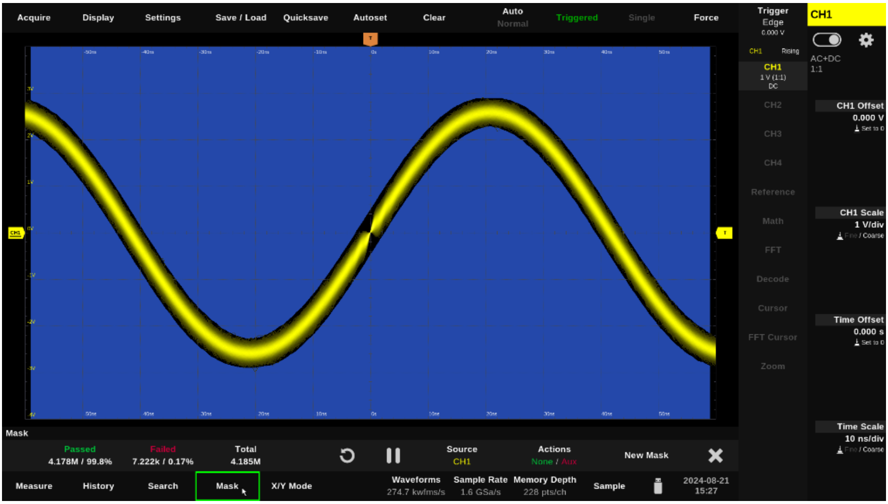
   The setting is made as a percentage of the screen verticals.
3. With the "Current" source setting, the recordings displayed with the last display update are searched. The "History" setting searches through all saved recordings.
4. As the search settings are structured in the same way as the trigger settings, you can use this function to copy the current settings from the trigger to the search dialog.
5. Copies the settings from the search dialog to the trigger settings.
6. The parameters associated with the search type can be configured in the "Specific" area.
7. The circular arrow can be used to set the default settings for the currently selected search type.
8. Closes the dialog.
9. Starts the search.

7.3.2. Search control bar

1. The time interval to the previous search hit.
2. The time of the search hit.
3. The time interval to the following search hit.
4. Reopens the Search configuration window.
5. Show the previous search hit.
6. The slider can be used to navigate manually between the search hits.
7. Show the following search hit.
8. Display of the number of the currently displayed search hit (top) and the number of search hits found (bottom). Tapping on the element opens an editor in which the desired number can be entered directly.
9. Ends the search mode.

7.4. Mask

You can use the mask test function to check waveforms for compliance with a specific progression during detection and perform actions such as outputting a sound signal and/or stopping detection.



To display the mask test control bar, please tap on "Mask" at the bottom of the screen. You can close it tapping again.

1. The number of recordings that could be checked against the mask without errors.
2. The number of recordings with errors.
3. The total number of recordings checked.
4. You can reset the counter values by tapping the circular arrow.
5. The pause symbol for pausing or the play symbol for starting the mask test run is displayed at this point.
6. Select one of the input channels for the mask check.
7. One or more actions can be defined that are to be executed after the mask test, depending on the result.
    a. Stop: Stops the acquisition of further measurement records and continues to display the tested waveform(s). The test can be resumed to acquire additional or new signal curves by pressing the Run button, for example.
    b. Aux: Triggers a pulse at the AUX OUT output.
    c. Quicksave: Saves the data, screenshots or more, depending on what was specified for the quicksave in the "Save / Load" menu.
    d. Sound: Outputs a sound signal.

8. "New Mask" opens the view for defining a new mask specification. If required, a single recording of the desired, correct signal curve should be created first. This curve can then be used as a template for the mask together with a deviation to be defined in the horizontal and vertical directions.
9. Closes the mask test function.

7.5. X/Y Mode

The XY display is a special type of diagram in which two signals are plotted against each other instead of one signal over time. In this display, one signal is shown on the X-axis and the other on the Y-axis.

The representation is particularly useful for analyzing phase relationships and Lissajous figures. Complex relationships between the two signals can be visualized and examined that would not be visible in a conventional time representation.

To activate the XY display, please tap on "X/Y Mode" at the bottom of the screen, set the channels in the dialog that then appears and tap on the slider to activate or deactivate the display.

7.6. Reference

The Reference function allows previously recorded signals to be saved as reference curves and displayed on the screen. These saved references can then be compared with current measurements. This is particularly useful for recognizing changes in the signal curve when comparing different circuit variants.

7.6.1. Creating new references

Tap on "Reference" in the control selection and then on the plus sign that appears to the left of it to create a new reference.

When creating new references, you can select the channel and a color in which the reference is to be displayed. When using several channels, it is advisable to select the channel color with a lower brightness for the reference.

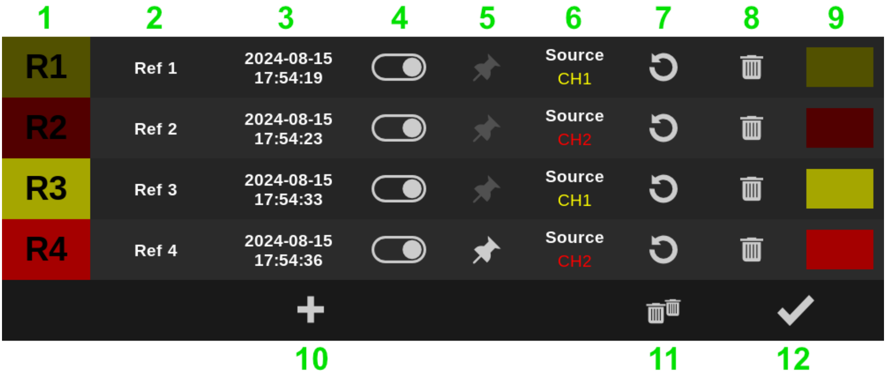
7.6.2. Control of the reference channels

You can move recorded references with an offset in the display height, scale them in height and move them in the X direction using a time offset.

To do this, you can use the markers, touch gestures or the settings area with the control dials. Select the reference beforehand by tapping the reference marker or using the control selection.

7.6.3. List of references

If references have already been created, these can be displayed in a tabular overview. To do this, tap on "Reference" in the control selection and then on the gearwheel that appears to the left of it.



1. Each reference has a number (R1, R2, ...) which is also displayed on the marker to the left of the measurement curve display area. You can adjust the order of the list by dragging the number.
2. You can assign individual names to all references. Depending on the setting made in the display menu, corresponding names will then be shown above the reference channel zero line permanently or when the offset is adjusted, only.
3. The date and time at which the reference was created.
4. Each reference display can be switched on and off using the slider.
5. The pin needle marks the currently selected reference. This determines which reference is currently displayed in the settings area and can be controlled using the rotary controls.
6. The source shows the channel from which the reference was created. The source can be changed at a later date, but the setting only becomes active after updating the reference.
7. Tapping the circular arrow updates the reference from the source.
8. The reference can be deleted by tapping the trash bin.
9. The display color of the reference. Tap to open the color selection window in which you can select a color.
10. Additional references can be created by tapping the plus sign. Up to eight references can be used.
11. All references are deleted by tapping the double trash bin.
12. Closes the window.

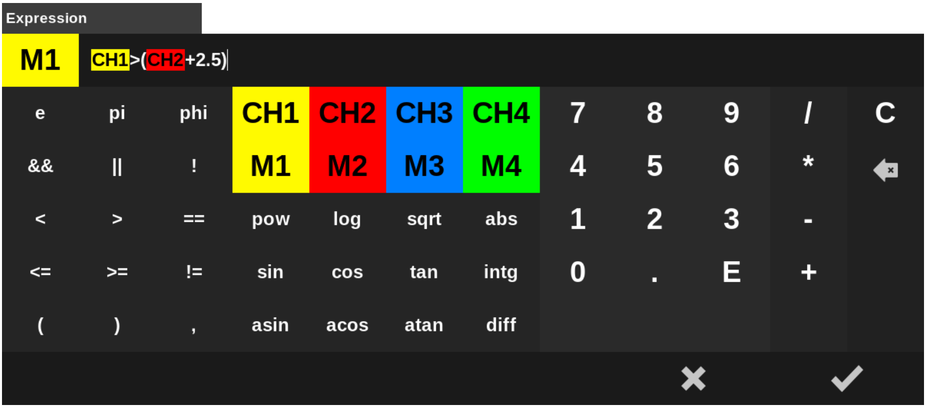
7.7. Math

You can use the Math channels to add, subtract, multiply and divide signal curves and perform more complex calculations. The results of these calculations are displayed as a separate signal curve.

Math channels are particularly useful, for example, when analyzing differential signals by subtracting two channels, to display the power consumption by multiplying the voltage measurement on one channel and the current measurement on a second channel or to carry out tests for special signal combinations of several channels using logical operators.

7.7.1. Creating new math channels and the formula editor

Tap on "Math" in the control selection and then on the plus sign that appears to the left of it to create a new math channel.



The formulas for each math channel can be entered freely in the formula editor. The rules of math (brackets, multiplication before addition, etc.) are adhered to.

Some examples:

- Basic arithmetic operations and constants: "CH1 * CH2 * pi"
  Channel 1 is multiplied
  by the second channel and pi.
- Exponentiation and root extraction: "pow(CH1,3) + sqrt(CH2)" Channel 1 is raised to the power of 3 and the square root of channel 2 is added to it.
- Logical operators: "CH1 > CH2 && CH1 < 2.5"
  The signal curve of the math channel goes to 1 at the points at which channel 1 has a greater value than channel 2 and channel 1 is below 2.5 (V).

The channels used in math channel formulas must be active in the measurement curve display area for the math channels to be updated.

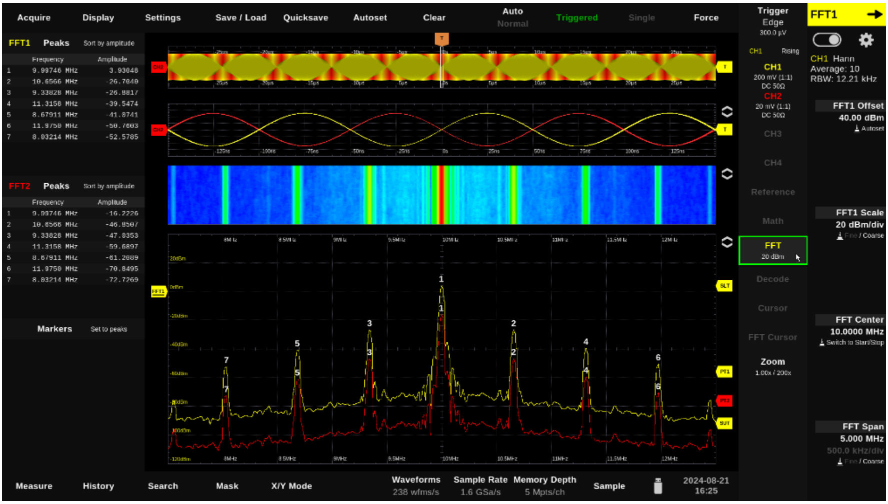
7.7.2. Listing of the math channels

If math channels have already been created, they can be displayed and adjusted in a tabular overview. To do this, tap on "Math" in the control selection and then on the gearwheel that appears to the left of it.

1. Each math channel has a designation (M1, M2, ...) which is also displayed on its marker to the left of the measurement curve display area. You can change the order of the list by dragging the number.
2. The formula used for the math channel. Tapping it opens the formula editor in which you can customize the formula.
3. The display of the math channel can be switched on and off using the slider.
4. The pin marks the currently selected math channel. This determines which math channel is currently displayed in the settings area and can be controlled using the rotary controls.
5. The unit selected for the display. V, A, W, Ω, Wb, C and Ws are available for selection.
6. The math channel can be deleted by tapping the trash bin.
7. The display color of the math channel. Tap to open the color selection window in which you can select a color.
8. Additional math channels can be created by tapping the plus sign. Up to four math channels can be used.
9. All math channels are deleted by tapping the double trash bin.
10. Closes the window.

7.8. Spectrum analysis and FFT

Signals can be broken down into their frequency components and analyzed using the fast Fourier transform (FFT for short). This function is ideal for determining the frequency components in noise and deducing potential causes for this noise. Furthermore, the quality and signal interference of an amplifier can be determined, for example.



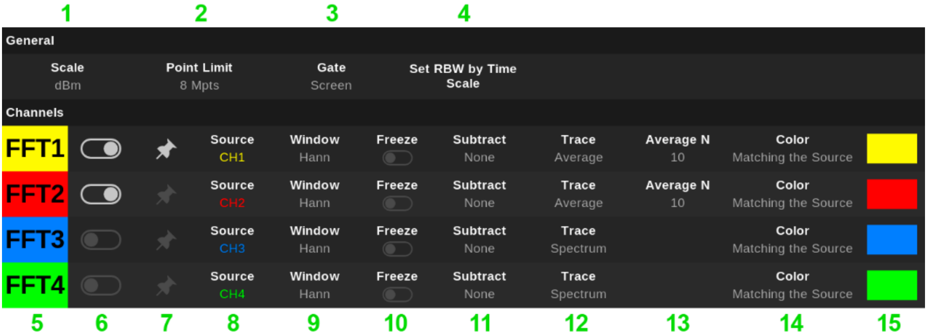
7.8.1. Display of the FFT waveform and the FFT settings window

Tap on "FFT" in the control selection and then on the gearwheel that appears to the left of it to open the FFT settings window.

Up to four FFT channels, two peak tables and a waterfall display can be used in parallel.

7.8.2. FFT settings window - the upper part

In the upper part of the FFT settings window you will find the general settings and the up to four FFT channels.



1. The vertical display can be in the logarithmic units dBm, dBV, dBmV and dBµV or in the linear unit V.
2. Up to eight million sampling points can be used for the FFT calculation. The more sampling points (measured values) are used, the finer the FFT frequency resolution (RBW). However, a large number of sampling points can lead to longer calculation time and a slower FFT result update rates.
3. Either the currently displayed range or the entire recording memory can be used for the analysis.
4. The time base required for a desired RBW (Resolution Bandwidth) can be automatically calculated by the software and set accordingly.
5. Each FFT channel has a designation (FFT1, FFT2, ...), which is also displayed on its marker to the left of the FFT display.
6. The display of the FFT channel can be switched on and off using the slider.
7. The pin needle marks the currently selected FFT channel. This determines which FFT channel is displayed in the settings area and can be controlled using the rotary controls.
8. The source shows the channel from whose measurement data the FFT is calculated. The source can be selected as required for each of the FFT channels and several FFTs can also be calculated from the data of one channel (e.g., to be able to display different window functions simultaneously).
9. The window functions determine how the measurement data is to be weighted, particularly at the edges of the recording. The following window functions are available: Rectangle, Hann, Hamming, Blackman, Flat Top, Gaussian, and Kaiser-Bessel. Each has its advantages and disadvantages and specific applications; please refer to the relevant literature if necessary.
10. When the FFT channel is frozen, it continues to be displayed but is no longer recalculated. This allows spectra to be compared before and after a circuit change, for example, by using additional FFT channels (similar to the references).

11. To normalize FFT curves or to filter out interfering components, the results of an FFT calculation can be subtracted from another FFT. Further information and application examples can be found in the section „Subtracting FFTs“.
12. Individual FFT spectra or an average of several FFT spectra can be displayed. Furthermore, the maximum or minimum value can be held for each frequency.
13. When the mean value display is activated, the number of FFT spectra over which the mean value is to be calculated can be specified.
14. The FFT channel color can be taken directly from the channel color or specified. can be displayed directly and separately or the display can be averaged over many other FFT calculations.
15. The display color of the FFT channel. Tap to open the color selection window in which you can select a color.

7.8.3. Subtracting FFTs

To normalize FFT curves or to filter out interfering components, the results of an FFT calculation can be subtracted from another FFT. As the functionality of the new FFT subtraction function is not necessarily obvious at first glance, here are two examples:

1. Example: Normalization of a measurement curve

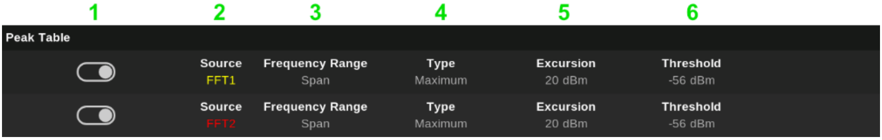
- Set FFT 1 to channel 1
- Also set FFT 2 to channel 1 and additionally activate „Subtract FFT 1“.
- FFT 2 now shows a flat line at 0 dB - logical, as both FFTs process identical signals with identical settings.
- If „Freeze“ is now activated for FFT 1, FFT 2 shows the normalized curve: All interference components and peaks contained in the frozen FFT 1 are removed.

2nd example: Removing ambient interference (far field)

- Set FFT 1 to channel 1 and activate „Subtract FFT 2“.
- Set FFT 2 to channel 2.
- Select e.g. an 8-fold average value for both FFTs.
- Connect a simple receiving antenna (e.g. a piece of wire in the BNC connector) to both channels - channel 1 close to the circuit to be tested, channel 2 at some distance.
- FFT 1 now predominantly shows the interfering peaks of the test circuit; broadband far-field interference, which is received with similar strength at both positions, is reliably suppressed by the subtraction.

7.8.4. Peak tables

In the center section of the FFT settings window you will find the settings for the peak tables.

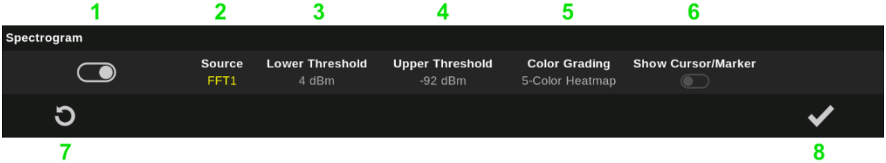


1. The peak table display can be switched on and off using the slider.
2. The source shows the FFT channel that is used for the peak table. The source can be selected as required for both tables and two tables can also be used for an FFT (e.g., to obtain a table for the complete frequency range and another table for the frequency range displayed in the FFT).
3. Peak detection can be limited to the frequency range displayed in the FFT (from start to stop) or over the entire frequency range.
4. Either the highest peaks (maximum) or the lowest peaks (minimum) can be listed.
5. The excursion setting defines the minimum height distance that a peak value in the frequency spectrum must have from its neighboring points in order to be recognized as an independent peak. This setting helps to ignore minor secondary peaks or noise and only list significant peaks in the peak table.
6. The threshold setting in the peak table defines the minimum amplitude value that a signal must reach in order to be listed in the peak table. This setting is useful for fading out unwanted, small peaks and concentrating on the relevant frequency peaks.

24211 Preetz, Handelsweg 16, Germany / Tel. +49 4342 90786-0 / service@batronix.com / www.batronix.com

7.8.5. Spectrogram (waterfall display)

In the lower part of the FFT settings window you will find the settings for the spectrogram display.



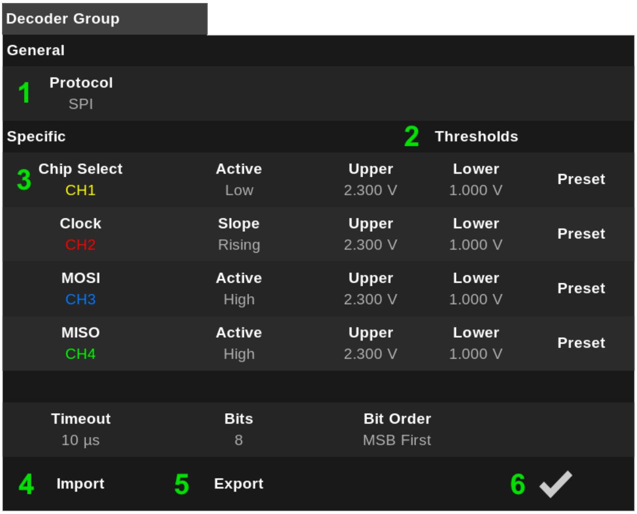
1. The spectrogram display can be switched on and off using the slider.
2. The source shows the FFT channel that is used for the spectrogram.
3. All levels up to the lower limit value are displayed in the "lowest" color of the selected color gradation.
4. All levels above the upper limit value are displayed in the "highest" color of the selected color gradation.
5. There are three color shades to choose from:
   a. 7-Color Heatmap: From colorless to blue, green, yellow, orange, red and white
   b. 5-Color Heatmap: From blue to green, yellow, and orange to red
   c. 2-Color Gradient: From blue to red
6. FFT cursors and FFT markers can also be displayed in the spectrogram.
7. The circular arrow can be used to reset the FFT data, mean values, and the spectrogram.
8. Close the window.

7.9. Decode

The decoder function converts the detected signals into the corresponding data packets or messages, which can then be further interpreted and analyzed.

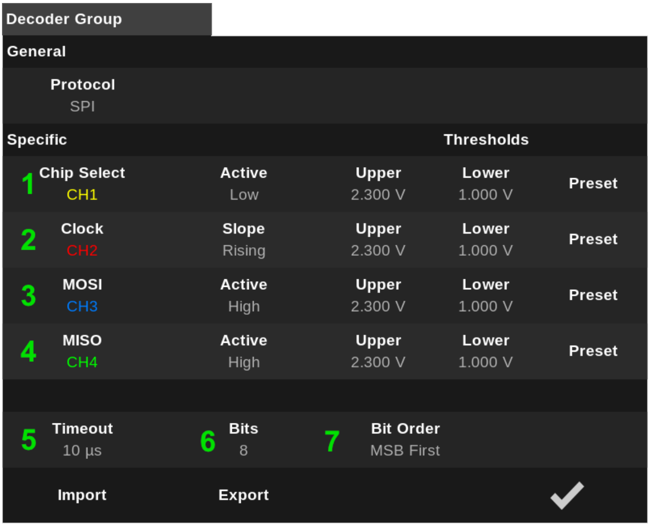
Tap on "Decoder" in the control selection and then on the plus sign that appears to the left of it to create a new decoder.

7.9.1. Decoder settings window



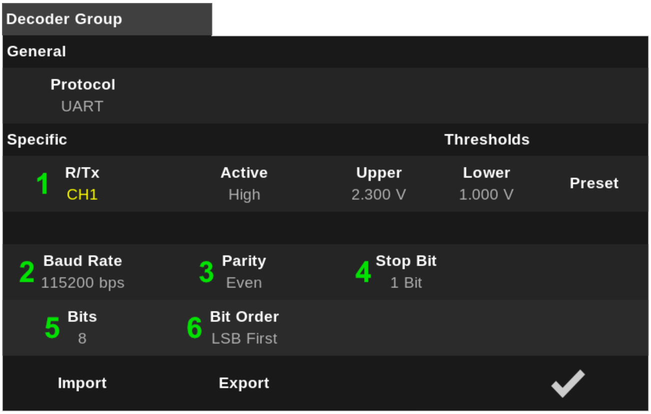
1. First set the appropriate protocol at the top under "Protocol".
2. The upper switching threshold defines the value starting from which the signal is to be recognized as a logical „high“ whereas the lower switching threshold correspondingly defines the values starting from which the signal is to be recognized as logical „low“. By selecting specific presets („Preset“), the threshold values can be set directly to the corresponding standard.
3. The setting options in the "Specific" area are adapted to the selected protocol. You will find the descriptions in the following chapters.
4. The assignment of individual signals can be imported automatically from the digital channel settings, provided that the standard abbreviations such as CS for "Chip Select" were used there when naming the digital channels.
5. The assignment made in this window can be exported to the digital channel settings and then set the channel names there accordingly.
6. Close the window.

7.9.2. SPI decoder



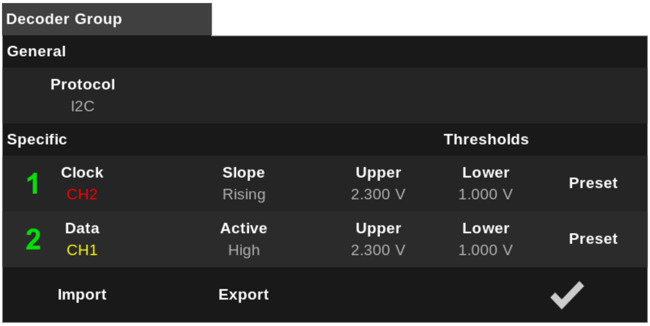
1. Set the signal source for the chip select signal. As an alternative to chip select, the data packet can also be recognized via timeout detection (see "5").
2. The clock pulse signal must be captured for decoding. Also set the appropriate signal edge with which the data is to be captured.
3. MOSI and MISO are the two data lines. You must have at least one of the two detected and decoded.
4. Like MOSI, only in the other data direction.
5. In addition to the CS signal, a timeout can be used to specify a time from which a new data packet is recognized.
6. The number of bits specifies how many bits are expected per data word.
7. The order of the bits can be set to MSB First or LSB First.
   MSB First stands for "Most Significant Bit" = most significant bit first
   LSB First stands for "Least Significant Bit" = least significant bit first

7.9.3. UART decoder



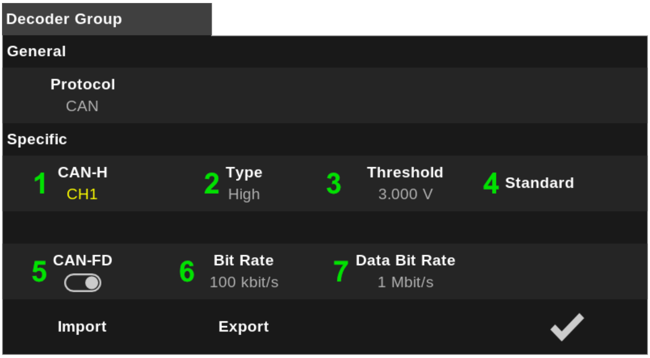
1. Set the signal source, the active level, and the switching thresholds.
2. The baud rate must be set appropriately for UART decoding in "bits per second" (bps).
3. The UART protocol has different parity settings that must be set appropriately here to recognize errors.
   a. None: No parity bit is used.
   b. Even: The parity bit is set so that the total number of "1" bits in the data packet (including the parity bit) is even.
   c. Odd: The parity bit is set so that the total number of "1" bits in the data packet (including the parity bit) is odd.
   d. Mark: The parity bit is always set to "1", regardless of the data. This setting is rarely used and does not provide any real error detection.
   e. Space: The parity bit is always set to "0", regardless of the data. Like mark parity, this setting is also rarely used and does not provide error detection.
4. The stop bit or stop bits signal the end of a data frame. This allows the receiver to recognize the end of the data packet and prepare for the next one.
5. The number of bits can be set from 4 to 32.
6. The order of the bits can be set to MSB First or LSB First.
   MSB First stands for "Most Significant Bit" = most significant bit first
   LSB First stands for "Least Significant Bit" = least significant bit first

7.9.4. I2C decoder



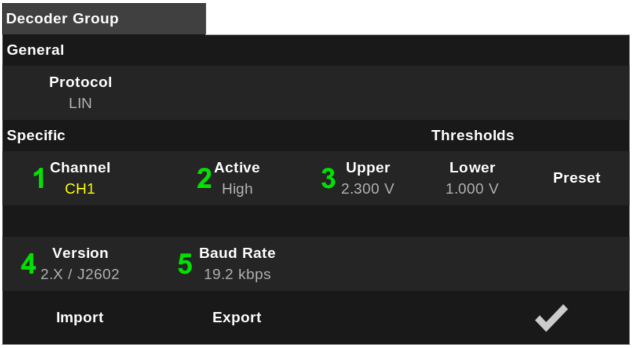
1. Set the signal source for the clock signal, the slope of the edges and the switching thresholds. With an I2C interface, the rising edges of the clock signal are usually used.
2. The data line of the I2C interface.

7.9.5. CAN(-FD) decoder



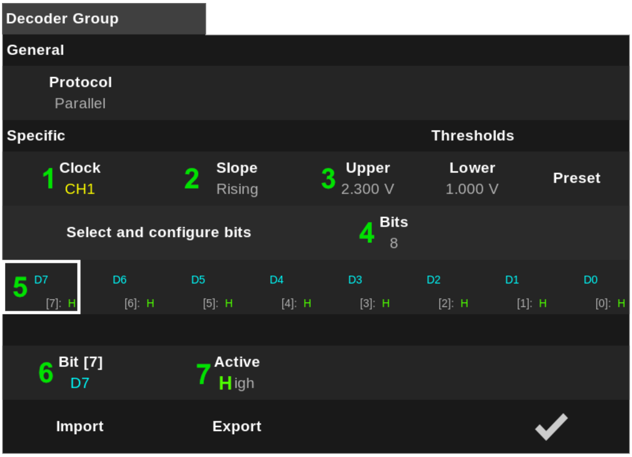
1. Set the signal source.
2. Decoding can take place on either the CAN-H or the CAN-L signal.
3. Set the switching threshold level manually or tap on "Standard".
4. You can select ISO 11898-2 or ISO 11898-3 to set the switching threshold levels appropriately.
5. Switch on the slider to perform CAN FD decoding.
6. The bit rate can be up to 10 Mbit/s.
7. With CAN-FD, the data part can have a different baud rate, which you can set here. The data bit rate can be up to 50 Mbit/s.

7.9.6. LIN decoder



1. Set the signal source.
2. The signal can be decoded as high-active or low-active.
3. Set the switching threshold manually or tap on "Logic type".
4. Select the appropriate LIN version (V1.X or V2.X / J2602).
5. The baud rate can be set at up to 8 Mbps.

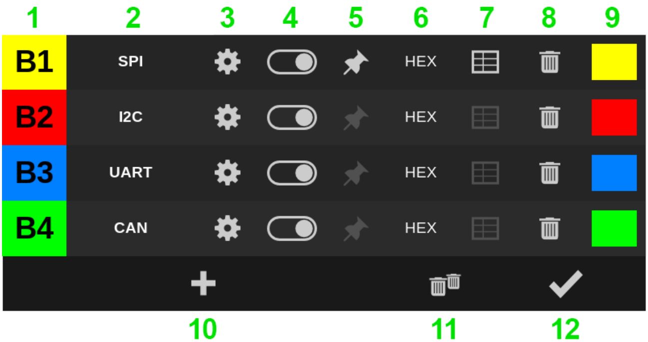
7.9.7. Parallel decoder



1. Decoding can take place synchronously based on a clock signal or asynchronously without a clock signal.
2. If you select an analog channel as the clock source, you can define here whether the data is transferred with a rising or falling edge.
3. Set the switching threshold level manually or tap on "Preset" to select from a predefined list of values.
4. Parallel interfaces with data widths between 1 to a maximum of 20 bits can be decoded.
5. Configure bit by bit here and further select bit source and any further bit-specific parameters under "6".
6. Set the source for the bit previously selected under "5". Up to four analog channels and, if available, up to 16 digital channels of the logic analyzer modules can be used at the same time.
7. For digital sources, choose between active high and active low bit interpretation or set the switching thresholds for analog sources.

7.9.8. List of decoders

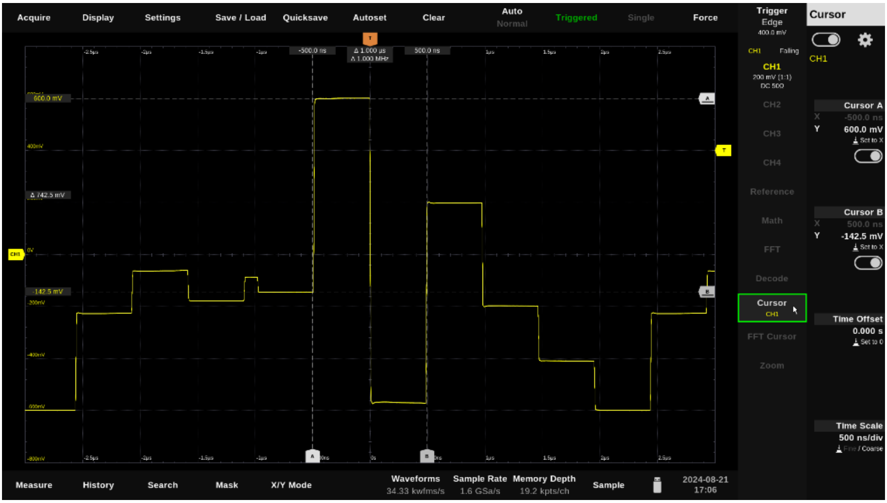
If decoders have already been created, they can be displayed and customized in a tabular overview. To do this, tap on "Decoders" in the control selection and then on the gearwheel that appears to the left of it.



1. Each decoder has a designation (B1, B2, ...) which is also displayed on its marker to the left of the measurement curve display area. You can adjust the order of the list by dragging the number.
2. Newly created decoders are initially given the name of the protocol. You can customize this name as required. This is particularly helpful if several decoders of the same protocol are used. Depending on the setting made in the display menu, the decoder name is then displayed when the marker is moved or permanently above the marker zero line.
3. The gearwheel opens the settings window for the decoder.
4. The decoder display can be switched on and off using the slider.
5. The pin marks the currently selected decoder. This determines which decoder is currently displayed in the settings area and can be moved in the display position using the rotary controls.
6. The decoded values can be displayed as a binary number (BIN), decimal number (DEC), hexadecimal number (HEX) or ASCII character (ASCII).
7. The decoded information can be displayed in a table.
8. The decoder can be cancelled by tapping the trash bin.
9. The display color of the decoder. Tap to open the color selection window in which you can select a color.
10. Additional decoders can be defined by tapping the plus sign. Up to four decoders can be specified and used at the same time.
11. Tapping the double trash bin cancels all decoders.
12. Closes the window.

7.10. Cursor

The cursor function can be used to carry out manual measurements on the signal display. The two horizontal cursors can be used to determine time points and time intervals, while the two vertical cursors can be used to determine levels and level differences.

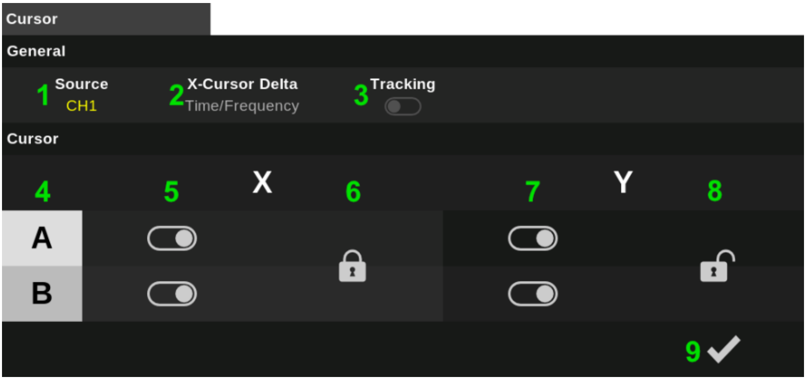


Tap on "Cursor" in the control selection to show the cursors. Tap on the gearwheel that appears to the left to open the cursor settings window.

You can hide the cursors again with a double tap on "Cursor".

You can move the cursors by touching the markers or by touching and dragging the lines.

7.10.1. Cursor settings window



1. A channel must be defined as the source so that the appropriate levels can be displayed.
2. With the X cursors, either the time, the frequency or both values can be displayed between the two cursors.
3. When the tracking function is switched on, the Y cursors follow the signal level at the position set with the corresponding X cursor.
4. In order to distinguish the cursor pairs, one is labelled with an A and the other with a B (also visible in the markers).
5. The display of the X cursor can be switched on and off using the slider.
6. When the lock function is activated, the distance between two cursors can be fixed. If one of the two cursors is moved, the second moves with it. The function can be switched on and off by tapping on it. If the function is activated, the lock is shown closed, otherwise it is shown open.
7. Corresponds to point 5, but here for the Y cursors.
8. Corresponds to point 6, but here for the Y cursors.
9. Closes the window.

7.11. FFT cursor and FFT marker

Either cursors or markers can be used in the FFT.

The FFT cursor function can be used to carry out manual measurements on the FFT display. The two horizontal cursors can be used to determine frequencies and frequency differences, while the two vertical cursors can be used to determine levels and level differences.

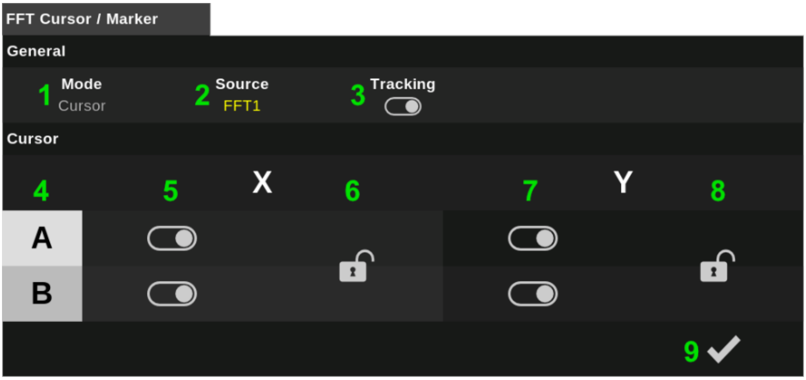
Up to ten markers can be set with the FFT marker function. The amplitude is displayed for each marker. The markers and the corresponding levels can be displayed in tabular form.

Tap on "FFT Cursor" or "FFT Marker" in the control selection to show the FFT cursors or markers. Tap on the gear wheel that appears to the left to open the settings window.

You can move the FFT cursors/markers by touching the markers or by touching and dragging the lines. You can hide these again with a double tap on "FFT Cursor/Marker".

7.11.1. FFT cursor settings window

In the control selection, tap on "FFT Cursor" or "FFT Marker" and then on the gearwheel that appears to the left of it to open the settings window.

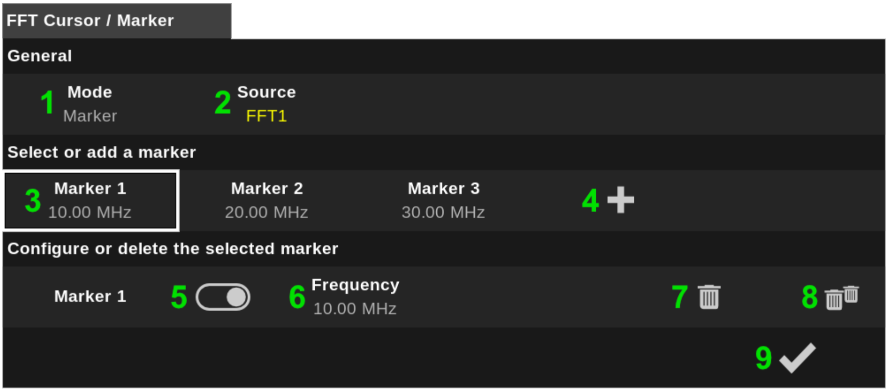


1. The mode setting can be used to switch between cursors and markers. The entry in the control selection will be displayed with the appropriate name.
2. An FFT must be defined as the source so that the appropriate levels can be displayed.
3. When the tracking function is switched on, the Y cursors follow the signal level at the frequency set with the corresponding X cursor.
4. To distinguish the cursor pairs, one is labelled with an A and the other with a B (also visible in the markers).
5. The display of the X cursor can be switched on and off using the slider.
6. When the lock function is activated, the distance between two cursors can be fixed. If one of the two cursors is moved, the second moves with it. The function can be switched on and off by tapping. If the function is activated, the lock is shown closed, otherwise it is shown open.
7. Corresponds to point 5, but here for the Y cursors.

8. Corresponds to point 6, but here for the Y cursors.
9. Closes the window.

7.11.2. FFT marker settings window

In the control selection, tap on "FFT Cursor" or "FFT Marker" and then on the gearwheel that appears to the left of it to open the settings window.



1. The mode setting can be used to switch between cursors and markers. The entry in the control selection is then also displayed with the appropriate name.
2. An FFT must be defined as the source so that the appropriate levels can be displayed.
3. Select a marker to switch it on or off or to set its frequency position.
4. Additional markers can be created using the plus sign. Up to 10 markers can be created and displayed.
5. The display of the marker previously selected under "4" can be switched on and off using the slider.
6. Tapping the configured frequency value opens a number editor in which it can be changed. The frequency can also be set using the rotary control when the number editor is open.
7. The marker can be deleted by tapping the trash bin.
8. All markers are deleted by tapping the double trash bin.
9. Closes the window.

7.12. Zoom

You can use the zoom view to display part of the measurement curve display in enlarged form. The magnification can be in the horizontal and/or vertical axis.

To switch on the zoom view, tap "Zoom" in the control selection. To switch it off, double tap it.

You can move the zoom view in the horizontal and vertical axes using the settings area or by touch and set the zoom factor in both axes.

For touch control, you can touch the zoom view with one finger and move it by dragging it vertically or horizontally. The zoom factor can be changed by touching the zoom view with two fingers and simultaneously spreading or merging them.

The zoomed area is displayed as a white rectangle in the "normal" measurement curve display. You can also touch and move this area there.

7.13. Troubleshooting - typical problems and solutions

7.13.1. No signal visible

Causes:

- Incorrect input channel selected
- Input deactivated (channel switched off)
- Incorrect time base or voltage range
- No or too infrequent trigger

Solutions:

- Activate active channel and set correctly
- Adjust time base and scaling
- Use „Auto setup“ or „Default settings“
- Check trigger source and level

7.13.2. Signal only appears sporadically or with a delay

Causes

- Trigger condition too narrow or incorrectly defined
- Trigger source not synchronized with the signal
- Single recording active

Solutions:

- Check trigger level and trigger type
- Assign trigger source correctly
- Switch to continuous recording (Run/Stop)

7.13.3. Measured values inaccurate or discontinuous

Causes:

- Incorrect probe factor or coupling
- No or incorrect calibration
- Insufficient probe contact / ground fault

Solutions:

- Set probe correctly (e.g. 10:1)
- Check or perform calibration
- Check ground connection

7.13.4. Device no longer starts or no longer starts completely

Causes:

- Software or settings problem

Solutions:

- Reset all settings when starting the device using the following procedure: Switch the device on as usual. As soon as the power LED lights up for the first time, press the Run/Stop button and the top rotary encoder at the same time. Keep both pressed until the boot screen appears.
- Please contact our support team. We will take care of it, of course such an error should not occur.

7.13.5. Device no longer responds / frozen

Causes:

- Software or display problem

Solutions:

- Restart via power button
- Load „Factory settings“
- Please contact our support team. We will take care of it, such an error should of course not occur.

7.13.6. No access to network / storage destination

Causes:

- Incorrect network configuration
- Path or rights problem with the network drive

Solutions:

- Check network settings
- Remount drive
- Check connection with IT department or support

7.13.7. Fan runs permanently / too early

Causes:

- Configuration for fan behavior changed in the „Settings“ menu

Solutions:

- Check/adjust setting in the „Settings“ menu

24211 Preetz, Handelsweg 16, Germany / Tel. +49 4342 90786-0 / service@batronix.com / www.batronix.com

7.13.8. Decoding does not work

Causes:

- Incorrect channels or level set
- Timing parameters not suitable

Solutions:

- Check decoder settings
- Compare protocol parameters with data sheet

24211 Preetz, Handelsweg 16, Germany / Tel. +49 4342 90786-0 / service@batronix.com / www.batronix.com