TELEDYNE LECROY
Everywhereyoulook™

Operator's Manual

HDO6000 / HDO6000A
High Definition
Oscilloscopes

HDO6000/HDO6000A High Definition Oscilloscopes Operator's Manual

October, 2020
hdo6000a-om-eng_01oct20.pdf

Contents

About This Manual

Thank you for purchasing a Teledyne LeCroy oscilloscope. We're certain you'll be pleased with the detailed features unique to our instruments.

This manual covers the operation and maintenance of all instruments in the MAUI HDO6000 series, although HDO6000A is shown here. With the introduction of later versions of the 64-bit MAUI software, particularly version 8.3 and later, the graphical user interface on some instruments looked very different from what was offered on earlier instruments and included different touch screen capabilities. Despite the difference in appearance, however, the functionality is the same unless otherwise stated. Where there are differences or limitations in capabilities, these are explained in the text.

Documentation for implementing COM Automation and remote control, and for using optional software packages sold for Teledyne LeCroy instruments can be downloaded from our free technical library at: teledynelecroy.com/support/techlib

Our website maintains the most current product specifications and should be checked for updates. Detailed specifications are listed on the product datasheet.

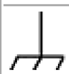
Note: Specifications are subject to change without notice.

Oscilloscope Overview and Set Up

Safety

Symbols

These symbols appear on the instrument or in documentation to alert you to important safety concerns:

	**Caution** of potential damage to instrument or **Warning** of potential bodily injury. Refer to manual. Do not proceed until the information is fully understood and conditions are met.
	**Caution**, high voltage; risk of electric shock or burn.
	**Caution**, contains parts/assemblies susceptible to damage by Electrostatic Discharge (ESD).
	Frame or chassis terminal (ground connection).
~	Alternating current.
	Standby power (front of instrument).

Precautions

Observe generally accepted safety procedures in addition to the precautions listed here. **The overall safety of any system incorporating this product is the responsibility of the assembler of the system.**

Use indoors only .

Use only within the operational environment listed. Do not use in wet or explosive atmospheres.

Maintain ground. The AC inlet ground is connected directly to the chassis of the instrument. To avoid electric shock, connect only to a mating outlet with a safety ground contact.

Caution: Interrupting the protective conductor inside or outside the oscilloscope, or disconnecting the safety ground terminal, creates a hazardous situation. Intentional interruption is prohibited.

Connect and disconnect properly. Do not connect/disconnect probes, test leads, or cables while they are connected to a live voltage source.

Observe all terminal ratings. Do not apply a voltage to any input that exceeds the maximum rating of that input. Refer to the body of the instrument for maximum input ratings.

Use only the power cord shipped with the instrument and certified for the country of use.

Keep product surfaces clean and dry. See Cleaning.

Do not remove the covers or inside parts. Refer all maintenance to qualified service personnel.

Exercise care when lifting. Use the built-in carrying handle.

Do not operate with suspected failures. Do not use the product if any part is damaged. Cease operation immediately and secure the instrumentfrom inadvertent use.

Operating Environment

Temperature:	5 °C to 40 °C.
Humidity:	Maximum relative humidity 90% up to 31 °C, decreasing linearly to 50% relative humidity at 40 °C.
Altitude:	Up to 10,000 ft (3,048 m) at or below 30 °C.

Measuring Terminal Ratings (C1-C4 and Ext)

Maximum Input Voltage:	50 Ω coupling ≤ 5 Vrms
	1 MΩ coupling ≤ 400 Vpk max. (Peak AC ≤ 10 kHz + DC) derating at 15 dB/decade from 10 kHz to 1.6 MHz, 10 Vpk max.above 1.6 MHz

Caution: Measuring terminals have no rated measurement category per IEC/EN 61010-1:2010. Measuring terminals are not intended to be connected directly to supply mains.

Cooling

The instrument relies on forced air cooling with internal fans and vents. Take care to avoid restricting the airflow to any part. In a benchtop configuration, leave a minimum of 15 cm (6 inches) around the sides between the instrument and the nearest object. The feet provide adequate bottom clearance. Follow rackmount instructions for proper rack spacing.

Caution: Do not block the cooling vents.

The instrument also has internal fan control circuitry that regulates the fan speed based on the ambient temperature. This is performed automatically after start-up.

Cleaning

Clean only the exterior of the instrumentusing a soft cloth moistened with water or an isopropyl alcohol solution. Do not use harsh chemicals or abrasive elements. Under no circumstances submerge the instrument or allow moisture to penetrate it. Dry thoroughly before connecting a live voltage source.

Caution: Unplug the power cord before cleaning. Do not attempt to clean internal parts.

Power

The instrument automatically adapts to the line voltage; manual voltage selection is not required.

AC Power Source:	100-240 VAC at 50/60 Hz (± 5%) 100-120 VAC at 400 Hz (± 5%)
Maximum Consumption:*	320 W (320 VA)
Nominal Consumption:	200 W (200 VA)
Standby Consumption:	4 W10 W

* All PC peripherals and active probes installed on four channels.

The provided power cords mate to a compatible power inlet on the instrument for making line voltage and safety ground connections. The AC inlet ground is connected directly to the chassis of the instrument. For adequate protection again electric shock, connect to a mating outlet with a safety ground contact.

Overview

Front of Oscilloscope

A. Touch screen display

B. Front panel

C. Stylus holder

D. Power button

E. Channel inputs (C1-4)

F. External trigger input

G. Mixed-Signal interface

H. Ground and Calibration output terminals

I. USB ports

J. Feet rotated back and tilted

Side of Oscilloscope

A. HDMI and DisplayPort ports for connecting external monitors
B. USB 3.1 Gen 1 ports (4)
C. Ethernet ports (2) for LAN connection or remote control
D. Mic and Speaker connection

Back of Oscilloscope

A. Built-in carrying handle
B. Auxiliary output
C. Ref In/Out for external reference clock
D. USBTMC port for remote control
E. AC power inlet

Front Panel

Front panel controls duplicate functionality available through the touch screen and are described here only briefly.

Knobs on the front panel function one way if turned and another if pushed like a button. The first label describes the knob's "turn" action, the second label its "push" action. Actions performed from the front panel always apply to the active trace.

Many buttons light to show the active traces and functions.

Trigger Controls

Level knob changes the trigger threshold level (V). The level is shown on the Trigger descriptor box. Pushing the knob sets the trigger level to the 50% point of the input signal.

READY indicator lights when the trigger is armed. **TRIG'D indicator** is lit momentarily when a trigger occurs.

Setup opens/closes the Trigger Setup dialog.

Auto sweeps after a preset time, even if the trigger conditions are not met.

Normal sweeps each time the trigger signal meets the trigger conditions.

Single sets Single trigger mode. The first press readies the oscilloscope to trigger. The second press arms and triggers the oscilloscope once (single-shot acquisition) when the input signal meets the trigger conditions.

Stop pauses acquisition. If you boot up the instrument with the trigger in Stop mode, a "No trace available" message is shown. Press the Auto button to display a trace.

Horizontal Controls

The **Delay knob** changes the Trigger Delay value (S) when turned. Push the knob to return Delay to zero.

The **Horizontal Adjust knob** sets the Time/division (S) of the acquisition system when the trace source is an input channel. The Time/div value is shown on the Timebase descriptor box. When using this control, the instrument allocates memory as needed to maintain the highest sample rate possible for the timebase setting. When the trace is a zoom, memory or math function, turn the knob to change the horizontal scale of the trace, effectively "zooming" in or out. By default, values adjust in 1, 2, 5 step increments. Push the knob to change to fine increments; push it again to return to stepped increments.

Math, Zoom, and Mem(ory) Buttons

The **Zoom** button creates a quick zoom for each open channel trace. Touch the zoom trace descriptor box to display the zoom controls.

The **Math** and **Mem**(ory) buttons open the corresponding setup dialogs.

If a Zoom, Math or Memory trace is active, the button illuminates to indicate that the Vertical and Horizontal knobs will now control that trace.

Vertical Controls

Offset knob adjusts the zero level of the trace (making it appear to move up/down relative to the center axis). The voltage value appears on the trace descriptor box. Push the knob to return Offset to zero.

Gain knob sets vertical scale (V/div). The voltage value appears on the trace descriptor box. By default, values adjust in 1, 2, 5 step increments. Push the knob to change to fine increments; push it again to return to stepped increments.

Channel (number) buttons turn on a channel that is off, or activate a channel that is already on. When the channel is active, pushing its channel button turns it off. A lit button shows the active channel.

Dig button enables digital input through the Digital Leadset on instruments with the Mixed Signal option.

Cursor Controls

Cursors identify specific voltage and time values on a waveform. The white cursor markers help make these points more visible. A readout of the values appears on the trace descriptor box. There are five preset cursor types, each with a unique appearance on the display. These are described in more detail in the Cursors section.

Type selects the cursor type. Continue pressing to cycle through all cursor until the desired type is found. The type "Off" turns off the cursor display.

Cursor knob repositions the selected cursor when turned. Push it to select a different cursor to adjust.

Adjust and Intensity Controls

The front panel **Adjust knob** changes the value in active (highlighted) data entry fields that do not have dedicated knobs. Pushing the Adjust knob toggles between coarse (large increment) or fine (small increment) adjustments.

When more data is available than can actually be displayed, the **Intensity** button helps to visualize significant events by applying an algorithm that dims less frequently occurring samples. This feature can also be accessed from the Display Setup dialog.

Miscellaneous Controls

Auto Setup performs an Auto Setup.

Default Setup restores the factory default configuration.

Print captures the entire screen and outputs it according to your Print settings. It can also be configured to output a LabNotebook entry.

Touch Screen enables/disables touch screen functionalilty.

Clear Sweeps resets the acquisition counter and any cumulative measurements.

Decode opens the Serial Decode dialog if you have serial data decoder options installed.

WaveScan opens the WaveScan dialog.

Spectrum opens the Spectrum Analyzer dialog if you have that option installed.

History opens the History Mode dialog.

Signal Interfaces

Teledyne LeCroy instruments offer a variety of interfaces to input analog or digital signals. See the oscilloscope product page at teledynelecroy.com for a list of compatible input devices.

ProBus Interface

Channel inputs C1-C4 utilize the ProBus interface. The ProBus interface contains a 6-pin power and communication connection and a BNC signal connection to the probe, with sense rings for detecting passive probes. It offers both 50 Ω and 1 MΩ input impedance and provides probe power and control for a wide range of probes such as high impedance passive probes, high impedance active probes, current probes, high voltage probes, and differential probes.

The ProBus interface completely integrates the probe with the channel. Upon connecting a Teledyne LeCroy probe, the probe type is recognized and some setup information, such as input coupling and attenuation, is performed automatically. This information is displayed on the Probe Dialog, behind the Channel (C*n*) dialog. System (probe plus instrument) gain settings are automatically calculated and displayed based on the probe attenuation.

The ProBus interface may have a BNC-terminated cable connected directly to it. Depending on the BNC connector used on the cable, the interface is rated for up to 4 GHz with 50 Ω coupling or 1 GHz with1 MΩ coupling.

Note: Operational bandwidth is equal to the maximum input frequency of your oscilloscope model. See the product datasheet.

Other Analog Inputs

EXT In can be used to input an external trigger pulse or sample clock signal.

REF In can be used to input an external reference clock signal.

These inputs have a simple BNC interface with no power supply. See your product datasheet for voltage and frequency ratings.

Mixed Signal Inputs

The digital leadset shipped with the -MS model oscilloscopes connects to the Mixed Signal Input on the front of the oscilloscope to input of up-to-16 lines of digital data. Physical lines can be preconfigured into different logical groups, Digital*n*, corresponding to a bus and renamed appropriately depending on the group. The transitions for each line may be viewed through different displays.

See Digital Setup Using the Digital Leadset for detailed instructions.

Probes

The oscilloscope is compatible with the included passive probes and most Teledyne LeCroy active probes that are rated for the instrument's bandwidth. Probe specifications and documentation are available at teledynelecroy.com/probes.

Passive Probes

The passive probes supplied are matched to the input impedance of the instrument but may need further compensation. Follow the directions in the probe instruction manual to compensate the frequency response of the probes.

If using other passive probes than those supplied, be sure to perform a low frequency calibration before using them to measure signal. You can use the signal from the Cal Out hook on the front of the oscilloscope.

Active Probes

Teledyne LeCroy offers a variety of active probes for use with your oscilloscope. Most active probes match probe to oscilloscope response automatically using probe response data stored in an on-board EEPROM. This ensures the best possible combined probe plus oscilloscope channel frequency response without the need to perform any de-embedding procedure.

Be aware that many active probes require a minimum oscilloscope firmware version to be fully operational. See the probe documentation.

Powering On/Off

Press the **Power button** to turn on the instrument.

The safest way to power down the oscilloscope is to use the **File > Shutdown** menu option, which will always execute a proper shut down process and preserve settings. Quickly pressing the Power button should also execute a proper shut down, but holding the Power button will execute a "hard" shut down (as on a computer), which we do not recommend doing because it does not allow the operating system to close properly, and setup data may be lost. Never power off by pulling the power cord from the socket, or by powering off a connected power strip or battery without first shutting down properly.

The Power button does not disconnect the instrument from the AC power supply. The only way to fully power down the instrument is to unplug the AC power cord.

We recommend unplugging the instrument if it will remain unused for a long period of time.

Caution: Do not change the instrument's Windows® Power Options setting from the default Never to System Standby or System Hibernate. Doing so can cause the system to fail.

Caution: Do not power on or calibrate with a signal attached.

Language Selection

To change the language of the oscilloscope application:

1. Go to **Utilities > Preference Setup > Preferences** and make a **Language** selection.
2. Follow the prompt to restart the application.

You can also select by touching the Language icon 

when it appears to the far right of the menu bar upon start up.

Connecting to Other Devices/Systems

Use the menu options listed below to configure connections to other devices.

LAN

The instrument is preset to accept a DHCP network address over a TCP/IP connection. Connect a cable from an Ethernet port on the side panel to a network access device. Go to **Utilities > Utilities Setup > Remote** to find the IP address.

To assign a static IP address, choose Net Connections from the Remote dialog. Use the standard Windows networking dialogs to configure the device address.

Choose **File > File Sharing** and open the Email & Report Settings dialog to configure email settings.

Audio/USB Peripherals

Connect the device to the appropriate port on the front or side of the instrument. These connections are "plug-and-play" and do not require any additional configuration.

Printer

MAUI oscilloscopes support USB printers compatible with the instrument's Windows OS. Go to **File > Print Setup** to configure printer settings. Select Properties to open the Windows Print dialog.

External Monitor

You may operate the instrument using the built-in touch screen or attach an external monitor for extended desktop operation. See your product datasheet for the supported monitor resolution.

Connect the monitor cable to a video output on the instrument. You can use an adaptor if the monitor cable has a different interface. Go to **Display > Display Setup > Open Monitor Control Panel** to configure the display. Be sure to select the instrument as the primary display.

To use the Extend Grids feature, configure the second monitor to extend, not duplicate, the oscilloscope display. If the external monitor is touch screen enabled, the MAUI user interface can be controlled through touch on the external monitor as well as the oscilloscope. See Windows 10 External Display Setup for additional instructions on setting up external monitors with Windows 10 oscilloscopes.

Note: The HDO, Zi series, and most 8-bit WaveSurfer and WaveRunner model oscilloscopes utilize Fujitsu touch-screen drivers. Because of conflicts, external monitors with Fujitsu drivers can not be used as touch screens, only as displays.

Remote Control

Go to **Utilities > Utilities Setup > Remote** to configure remote control. Connect the oscilloscope to the network/controller using the cable type required by your selection.

- VICP(TCP/IP) and VXI-11(LXI) over Ethernet are supported standard, as is USBTMC.
- GPIB is supported with the use of the optional USB-GPIB adapter.

Note: If using TCP/IP, you can connect over the LAN or directly to the controller. Be sure the oscilloscope is on the same subnet as the controller. Depending on the host, you may need to use a cross-over cable to make a direct connection.

Sample Clock

To input an external sample clock signal, connect a BNC cable from the EXT input to the other device. Go to Timebase > Horizontal Setup > Clock Source to configure the clock.

Reference Clock

To input/output a reference clock signal, connect a BNC cable from the Ref In/Out connector to the other instrument. Go to **Timebase > Horizontal Setup > Clock Source** to configure the clock.

Auxiliary Output

To output signal from the instrument to another device, connect a BNC cable from Aux Out to the other device. Go to **Utilities > Utilities Setup > Aux Output** to configure the output.

Oscilloscope Application

Waveform rendering and measurement functions are performed by the MAUI oscilloscope software, which runs on the Microsoft Windows platform. MAUI, the Most Advanced User Interface, forms the front-end of Teledyne LeCroy oscilloscopes, providing a single interface for all standard and optional oscilloscope applications.

The oscilloscope software (firmware and standard applications) is active upon delivery. At power-up, the instrument loads the software automatically.

Free updates are available periodically from the software download page of our website at teledynelecroy.com. Go to Oscilloscope Downloads > Firmware Upgrade.

Registered users can receive an email notification when a new update is released. Follow the instructions in Firmware Update to download and install the software.

If you decide to purchase an option, you will receive a license key via email that activates the optional features. See Options for instructions on activating optional software packages.

Customization

The MAUI application is a COM Automation server. All the configurable application objects that are presented through MAUI can be controlled using COM Automation strings embedded in remote control scripts, or sent via remote utilities such as the free WaveStudio software. See the *MAUI Oscilloscopes Remote Control and Automation Manual* for instructions.

Besides reading waveform and measurement data from the oscilloscope, a common use of Automation is the creation of remote setup files. In fact, MAUI provides a simple way to save any oscilloscope configuration to a LeCroy System Setup (.LSS) file, which is nothing more than a COM Automation program written in VB Script, ready to restore the entire saved configuration when executed.

Our proprietary LabNotebook feature goes even further to save not only the setups but also the waveform data to a file that can restore the full oscilloscope display to the exact state in which it was saved.

Further customization, such as the introduction of custom math or measurement processors into MAUI, can be performed on oscilloscopes equipped with CustomDSO. See our website for examples.

Volatile vs. Non-Volatile Settings

Most of the oscilloscope settings are volatile, meaning they will automatically revert to the factory default whenever the oscilloscope is rebooted, or when you choose to recall the default setup using the front panel Default Setup button or the Recall dialog.

Those settings that are the exception to this rule are called *non-volatile*. These settings will be retained session to session until you manually change them. Non-volatile settings include:

- All preferences settings (including acquisition, calibration, color and miscellaneous)
- All networking, remote control and email settings
- All printer settings and screen image (file) preferences
- All report settings, including logo selections
- All file paths and names, including auto-naming selections

Using MAUI

MAUI (Most Advanced User Interface) is Teledyne LeCroy's unique oscilloscope user interface. MAUI provides an extensible front-end to the MAUI oscilloscope application, integrating software options into a single application that can be controlled using the latest Windows touch screen features.

Touch Screen

With MAUI, the touch screen is the principal control center of the oscilloscope. The entire display is active: use your finger or a stylus to touch, drag, swipe, and draw selection boxes. With the addition of OneTouch, gestures such as pinch and flick can also be used to control the screen.

Many controls that display information also work as “buttons” to access other functions, and even the waveform traces can be manipulated. If you have a mouse installed, you can click anywhere you can touch to activate a control; in fact, you can alternate between clicking and touching, whichever is convenient for you.

The touch screen is divided into the following major control groups:

Menu Bar

The top of the window contains a complete menu of functions. Making a selection here changes the dialogs displayed at the bottom of the screen. While many operations can also be performed from the front panel or launched via the descriptor boxes, the menu bar is the best way to access dialogs for Save/Recall (File) functions, Display functions, Status, LabNotebook, Pass/Fail setup, optional Analysis packages, and Utilities/Preferences setup.

You will see a **Q-Scape** 

control added to the menu bar when the Q-Scape Multi-tabbed Display features are available. This allows you to choose different tabbed display configurations.

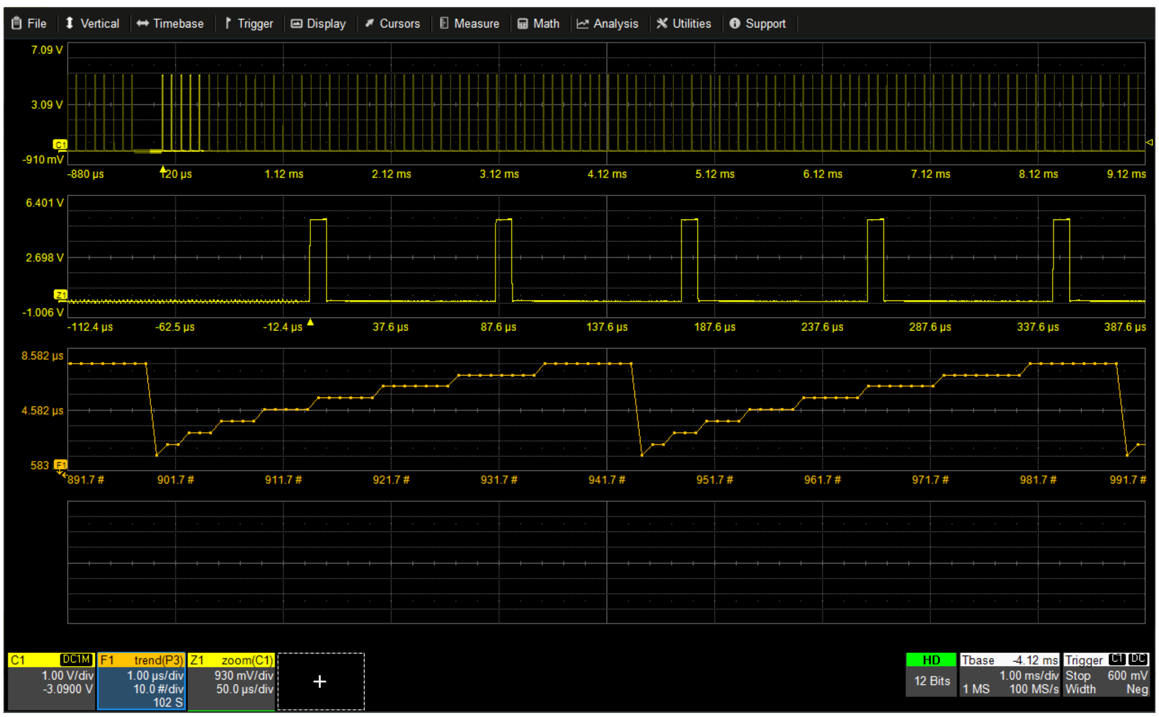
If an action can be "undone", the **Undo** button restores the oscilloscope to the prior state.

Grid

The grid displays the waveform traces. Every grid is 8 Vertical divisions representing the full number of 4096 Vertical levels and 10 Horizontal divisions each representing acquisition time. The value represented by Vertical and Horizontal divisions depends on the Vertical and Horizontal scale of the traces that appear on the grid.

Multi-Grid Display

The screen can be divided into multiple grid configurations, each grid showing different types and numbers of traces (in Auto Grid mode, it will divide automatically as needed). Regardless of the number and orientation of grids, every grid always represents the same number of Vertical levels. Therefore, absolute Vertical measurement precision is maintained.See Display.



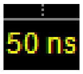
Different types of traces opening in a multi-grid display.

Q-Scape Tabs

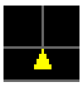
On instruments with the Q-Scape multi-tabbed display features, Tabs 1-4 appear above the grids. Each tab is a distinct multi-grid display. See Display for more information about using Q-Scape.

Grid Indicators

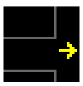
These indicators appear around or on the grid to mark important points on the display. They are matched to the color of the trace to which they apply. When multiple traces appear on the same grid, indicators refer to the *foreground* trace—the one that appears on top of the others.



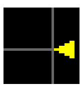
Axis labels mark the times/units represented by a grid division. They update dynamically as you pan the trace or change the Vertical/Horizontal scale. Originally shown in absolute values, the labels change to show delta from 0 (center) when the number of significant digits grows too large. The number of labels that appear on each grid depends on the total number of grids open. To remove axis labels, go to Display > Display Setup and deselect Axis Labels.



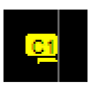
Trigger Time, a small triangle along the bottom (horizontal) edge of the grid, shows the time of the trigger. Unless Horizontal Delay is set, this indicator is at the zero (center) point of the grid. Delay time is shown at the top right of the Timebase descriptor box.



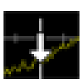
Pre/Post-trigger Delay, a small arrow to the bottom left or right of the grid, indicates that a pre- or post-trigger Delay has shifted the Trigger Position indicator to a point in time not displayed on the grid. All Delay values are shown on the Timebase Descriptor Box.



Trigger Level at the right edge of the grid tracks the trigger voltage level. A solid triangle indicates the last triggered level. If you change the trigger level prior to acquisition (e.g., while in Stop mode), a hollow triangle of the same color appears at the new trigger level. The trigger level indicator is not shown if the triggering channel is not displayed.



Zero Volts Level is located at the left edge of the grid. One appears for each open trace on the grid, sharing the number and color of the trace.



Cursor markers appear over the grid to indicate the voltage and time being measured on the waveform. Drag-and-drop cursor markers to quickly reposition them.

Grid Intensity

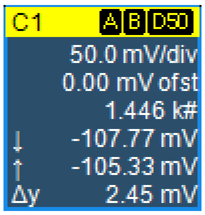
You can adjust the brightness of the grid lines by going to **Display > Display Setup** and entering a new **Grid Intensity** percentage. The higher the number, the brighter and bolder the grid lines.

Descriptor Boxes

Trace descriptor boxes appear just beneath the grid whenever a trace is turned on. They function to:

- Inform—descriptors summarize the current trace settings and its activity status.
- Navigate—touch the descriptor box once to activate the trace, again to open the setup dialog.
- Arrange—drag-and-drop descriptor boxes to move traces among grids.
- Configure—drag-and-drop descriptor boxes to change source or copy setups.

Trace Descriptor Box



Channel trace descriptor boxes correspond to analog signal inputs. They show (clockwise from top left): Channel Number, Pre-processing list, Coupling, Vertical Scale (gain) setting, Vertical Offset setting, Sweeps Count (when averaging), Vertical Cursor positions, and Number of Segments acquired (when in Sequence mode).

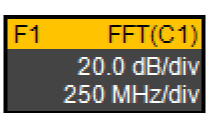
If you are interleaving channels, channel descriptor boxes will show the channel's overall acquisition status: trigger only, active, or not active.

Codes are used to indicate coupling and other processes affecting the channel. The short form is used when several processes are in effect.

Symbols on Descriptor Boxes

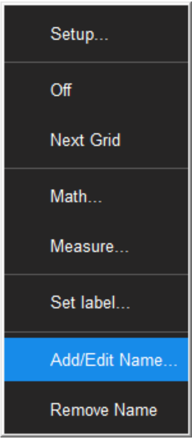
Processing Type	Long Form	Short Form
Coupling	DC50, DC1M, AC1M or GND	D50, D1, A1 or G
Bandwidth Limiting	BWL	B
Averaging	AVG	A
(Sinx)/x Interpolation / Enhanced Sample Rate	SINX / ESR	S / E
Deskew	DSQ	DQ
Noise Filter (ERes)	FLT	F
Inversion	INV	I

Note: When (Sinx)/x interpolation is applied by the Enhanced Sample Rate feature, the C*n* descriptor box will show ESR.



Similar descriptor boxes appear for math (F*n*), zoom (Z*n*), and memory (M*n*) traces, also sensor (SE*n*) traces if your system includes a SAM40 Sensor Acquisition Module. These descriptor boxes show any Horizontal scaling that differs from the signal timebase. Units will be automatically adjusted for the type of trace.

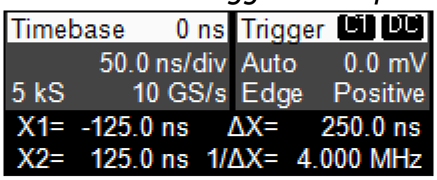
Trace Context Menu



The trace context menu is a pop-up menu of actions to apply to a trace, such as turn off, move to next grid, measure, label or rename.

To open the trace context menu, touch and hold on the trace descriptor box until a white box appears (or right-click with a mouse).

Timebase and Trigger Descriptor Boxes



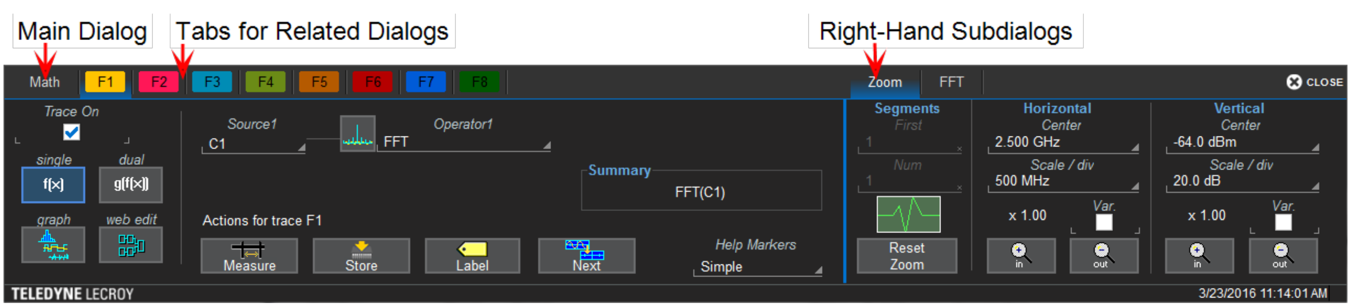
The Timebase descriptor box shows: (clockwise from top right) Horizontal Delay, Time/div, Sample Rate, Number of Samples and Sampling Mode (blank when in real-time mode).

The Trigger descriptor box shows: (clockwise from top right) Source and Coupling, Level (V), Slope/Polarity, Type and Mode.

Horizontal (time) cursor readout, including the time between cursors and the frequency, is shown beneath the TimeBase and Trigger descriptor boxes. See the Cursors section for more information.

Dialogs

Dialogs appear at the bottom of the display for entering setup data. The top dialog will be the main entry point for the selected functionality. For convenience, related dialogs appear as a series of tabs behind the main dialog. Touch the tab to open the dialog.



Right-Hand Subdialogs

At times, your selections will require more settings than can fit on one dialog, or the task invites further action, such as zooming a new trace. In that case, subdialogs will appear to the right of the dialog. These subdialog settings always apply to the object that is being configured on the left-hand dialog.

Action Toolbar

Several setup dialogs contain a toolbar at the bottom of the dialog. These buttons enable you to perform commonplace tasks—such as turning on a measurement—without having to leave the underlying dialog. Toolbar actions always apply to the active trace.

Measure opens the Measure pop-up to set measurement parameters on the active trace.

Zoom creates a zoom trace of the active trace.

Math opens the Math pop-up to apply math functions to the active trace and create a new math trace.

Decode opens the main Serial Decode dialog where you configure and apply serial data decoders and triggers. This button is only active if you have serial data software options installed.

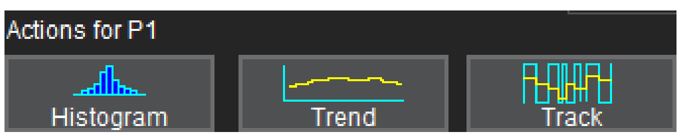
Store loads the active trace into the corresponding memory location (C1, F1 and Z1 to M1; C2, F2 and Z2 to M2, etc.).

Find Scale performs a vertical scaling that fits the waveform into the grid.

Add/Edit Name opens the virtual keypad for you to alias the trace.

Label opens the Label pop-up to annotate the active trace.

Histogram, **Trend**, and **Track** buttons appear in the Action toolbar at the bottom of the **Parameter** (P*n*) dialogs. They allow you to create a Math function to plot the parameter while remaining on the measurement dialogs.



Message Bar

At the bottom of the oscilloscope display is a narrow message bar. The current date and time are shown at the far right. Status, error, or other messages are shown at the far left, where "Teledyne LeCroy" normally appears.

You will see the word "Processing..." highlighted with red at the right of the message bar when the oscilloscope is processing your last acquisition or calculating.

This will be especially evident when you change an acquisition setting that affects the ADC configuration while in Normal or Auto trigger mode, such as changing the Vertical Scale, Offset, or Bandwidth. Traces may briefly disappear from the display while the oscilloscope is processing.

MAUI with OneTouch

Gestures like touch, drag, swipe, pinch and flick can be used to create and change setups with one touch. Just as you change the display by using the setup dialogs, you can change the setups by moving different display objects. Use the setup dialogs to refine OneTouch actions to precise values.

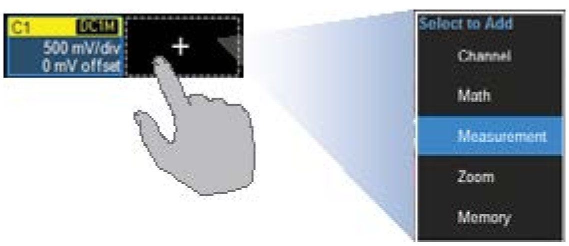
As you drag & drop objects, valid targets are outlined with a white box. When you're moving over invalid targets, you'll see the "Null" symbol (Ø) under your finger tip or cursor.

Turn On

To **turn on a new channel, math, memory, or zoom trace**, drag any descriptor box of the same type to the Add New ("+") box. The next trace in the series will be added to the display at the default settings. It is now the active trace.

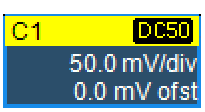
If there is no descriptor box of the desired type on the screen to drag, touch the Add New box and choose the trace type from the pop-up menu.

To **turn on the Measure table** when it is closed, touch the Add New box and choose Measurement.



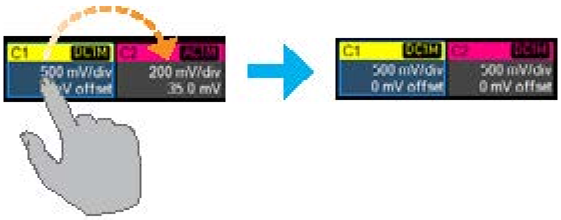
Activate

Touch a trace or its descriptor box to *activate* it and bring it to the *foreground*. When the descriptor box appears highlighted in blue, front panel controls and touch screen gestures apply to that trace.

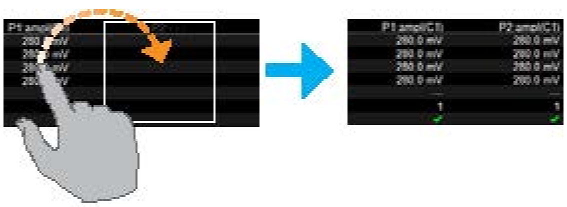


Copy Setups

To **copy the setup of one trace** to another of the same type (e.g., channel to channel, math to math), drag-and-drop the source descriptor box onto the target descriptor box.

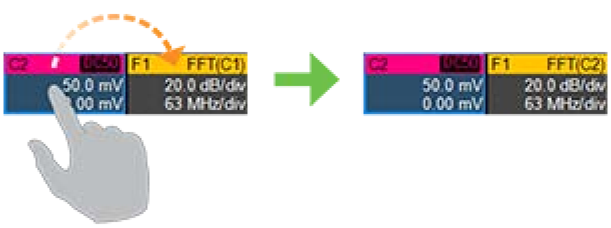


To **copy the setup of a measurement** (P*n*), drag-and-drop the source column onto the target column of the Measure table. You can do the same for a query (Q*n*) on the Pass/Fail readout table.

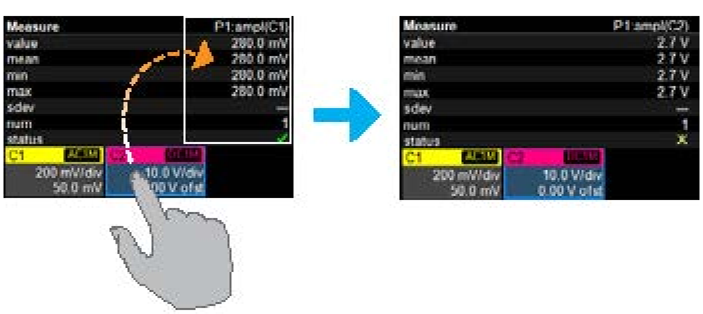


Change Source

To **change the source of a trace**, drag-and-drop the descriptor box of the desired source onto the target descriptor box. You can also drop it on the Source field of the target setup dialog.

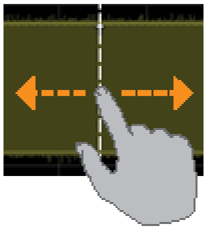


To **change the source of a measurement**, drag-and-drop the descriptor box of the desired source onto the parameter (P*n*) column of the Measure table. You can do the same to a query (Q*n*) on the Pass/Fail readout table.

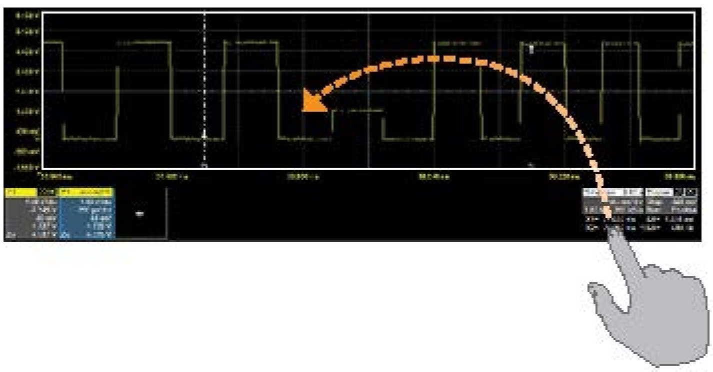


Position Cursors

To **change cursor measurement time/level**, drag cursor markers to new positions on the grid. The cursor readout will update immediately.

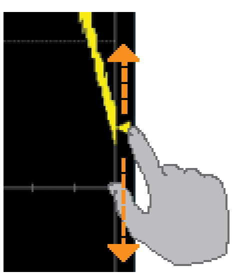


To **place horizontal cursors** on zooms or other calculated traces where the source Horizontal Scale has forced cursors off the grid, drag the cursor readout from below the Timebase descriptor to the grid where you wish to place the cursors. The cursors are set at either the 5 (center) or the 2.5 and 7.5 divisions of the grid. Cursors on the source traces adjust position accordingly.

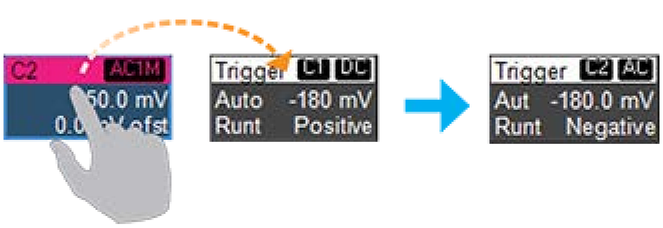


Change Trigger

To **change the trigger level**, drag the Trigger Level indicator to a new position on the Y axis. The Trigger descriptor box will show the new voltage Level.

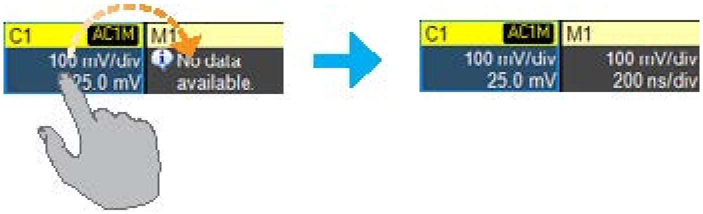


To **change the trigger source channel**, drag-and-drop the desired channel (C*n*) descriptor box onto the Trigger descriptor box. The trigger will revert to the coupling and slope/polarity last set on that channel.



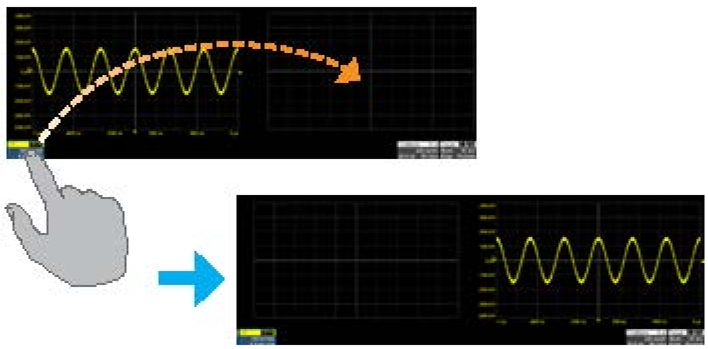
Store to Memory

To **store a trace to internal memory**, drag-and-drop its trace descriptor box onto the target memory (M*n*) descriptor box.



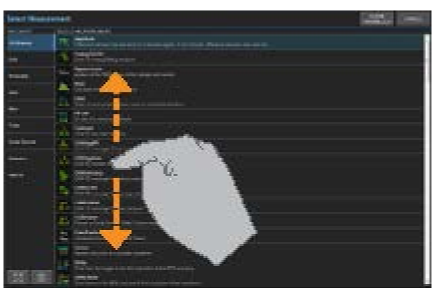
Move Trace

To **move a trace to a different grid**, drag-and-drop the trace descriptor box onto the target grid. If you are in a multi-tabbed Q-Scape display mode, you can move traces between tabs this way.



Scroll

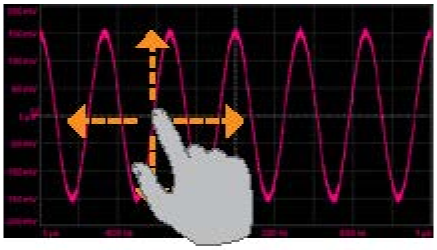
To **scroll long lists of values** or readout tables, swipe the selection dialog or table in an up or down direction.



Pan/Swipe Trace

To **pan a trace**, activate it to bring it to the forefront, then drag the waveform trace right/left or up/down. If it is the source of any other trace, that trace will move, as well.

For channel traces, the Timebase descriptor box will show the new Horizontal Delay value. For other traces, the zoom factor controls show the new Horizontal Center.



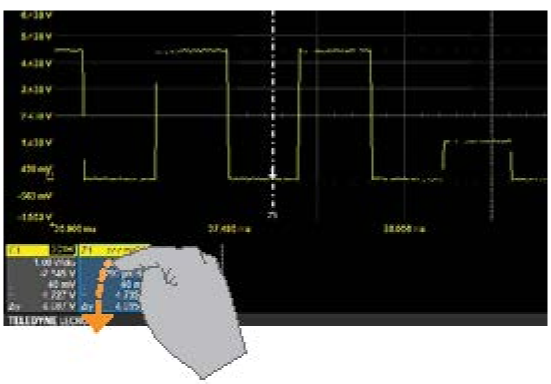
Tip: If you are using the multi-zoom feature, all time-locked traces will pan together.

To pan at an accelerated rate, **swipe** the trace right/left or up/down.

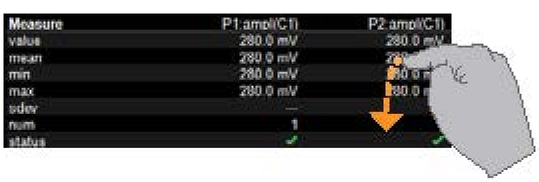
Swipe from the far left of the screen to switch "tabs" and show the Windows desktop. Swipe from left again to return to the oscilloscope application.

Turn Off

To **turn off a trace**, flick the trace descriptor box toward the bottom of the screen.

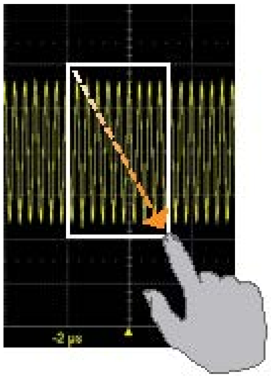


To **turn off a measure parameter** or Pass/Fail query, flick the P*n* or Q*n* cell toward the bottom of the screen. If it's the last active cell of the table, the table will close.

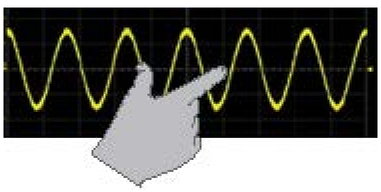


Zoom

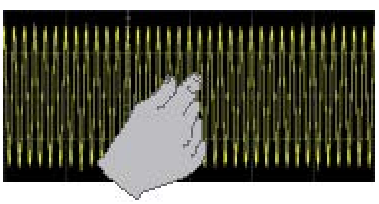
To **create a new channel zoom trace**, touch then drag diagonally to draw a rectangle around the portion of the trace you want to zoom. Touch the Z*n* descriptor box to open the zoom factor controls and adjust the zoom exactly.



To **"zoom in"** on any trace, unpinch two fingers over the trace horizontally.



To **"zoom out"** on any trace, pinch two fingers over the trace horizontally.



> **Note:** Pinch gestures do not create a separate zoom (Z*n*) trace, they only adjust the Horizontal Scale. When you pinch a channel (C*n*) trace, the Timebase for all channels changes. If the trace is the source of any other, all its dependent traces change, as well.

New Windows 10 Gestures

Windows 10 oscilloscopes support new touch screen gestures for accessing applications and settings:

- Swipe from the left of the screen to switch applications or show the desktop. Continue swiping to return to MAUI.
- Swipe from the right of the screen to open the Action Center, from where you can access system settings.

Controlling Traces

Traces are the visible representations of waveforms that appear on the display grid. They may show live inputs (C*n*, Digital*n*, SE*n*), a math function applied to a waveform (F*n*), a stored memory of a waveform (M*n*), a zoom of a waveform (Z*n*), or the processing results of special analysis software.

Traces are a touch screen object like any other and can be manipulated. They can be panned, moved, labeled, zoomed and captured in different visual formats for printing and reporting.

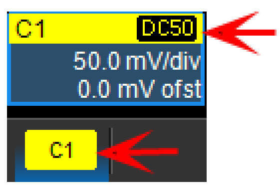
Each visible trace will have a descriptor box summarizing its principal configuration settings. See OneTouch Help for more information about how you can use traces and trace descriptor boxes to modify your configurations.

Active Trace

Although several traces may be open, only one trace is *active* and can be adjusted using front panel controls and touch screen gestures. A highlighted descriptor box indicates which trace is active. All actions apply to that trace until you activate another. Touch the trace descriptor box to make it the active trace (and the foreground trace in that grid).

Active trace descriptor (left), inactive trace descriptor (right).

Whenever you activate a trace, the dialog at the bottom of the screen automatically switches to the appropriate setup dialog.



Active descriptor box matches active dialog tab.

Foreground Trace

Since multiple traces can be opened on the same grid, the trace shown on top of the others is the *foreground* trace. Grid indicators (matched to the input channel color) represent the foreground trace.

Touch a trace or its descriptor box to bring it to the foreground. This also makes it the active trace.

Note that a *foreground* trace may not be the same as the *active* trace. A trace in a separate grid may subsequently become the active trace, but the indicators on a given grid will still represent the foreground trace in that group.

Turning On/Off Traces

Turn On/Off Analog Trace

To turn on a channel trace, do any of the following:

- From the front panel, press the **Channel button**.
- From the touch screen, choose **Vertical > Channel *x* Setup**.
- Touch the **Add New box** and select **Channel**, or drag another Channel (C*n*) descriptor box to the Add New box.

To turn off a trace, press the front panel Channel button a second time, or from the touch screen, do any of the following:

- If the instrument has OneTouch, flick the trace descriptor box toward the bottom of the screen.
- Right click on the descriptor box and choose **Off**.
- Touch the descriptor box and clear the **Trace On** checkbox on the setup dialog.

Turn On/Off Digital Trace

From the touch screen, choose **Vertical > Digital*n* Setup** then check **Group** on the Digital*n* dialog.

Turn off the trace, clear the Group checkbox.

Turn On/Off Zoom Trace

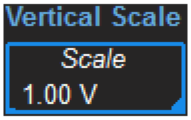
See Creating Zooms.

Turn On/Off Other Trace

To turn on/off math or memory traces, check or clear the Trace On box on the respective setup dialogs.

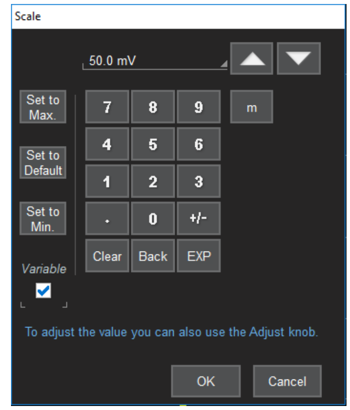
You can also touch the **Add New box** and select the trace type, or drag another descriptor box of that type to the Add New box (e.g., drag M1 to Add New to turn on a the next available memory trace).

Adjusting Traces



To adjust Vertical Scale and Offset, or Horizontal Scale and Delay, just activate the trace and use the front panel knobs. To make other adjustments—such as units—touch the trace descriptor box twice to open the appropriate setup dialog.

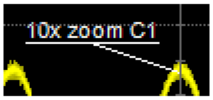
Many settings can be adjusted by selecting from the pop-up that appears when you touch a control. When an entry field appears highlighted in blue after touching, it is *active* and can be adjusted by turning the front panel knobs. Fields that don't have a dedicated knob (as do Vertical Level and Horizontal Delay) can be modified using the Adjust knob.



If you have a keyboard installed, you can type entries in an active (highlighted) data entry field. Or, you can touch again, then "type" the entry by touching keys on the virtual keypad or keyboard.

To use the virtual keypad, touch the soft keys exactly as you would a calculator. When you touch OK, the calculated value is entered in the field.

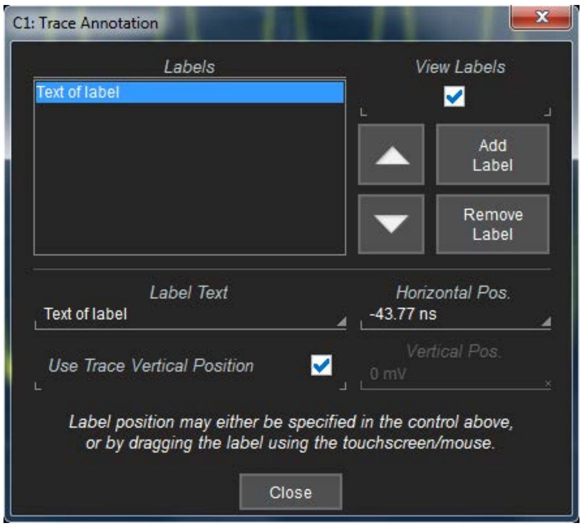
Labeling Traces



The Label function gives you the ability to add custom annotations to the trace display. Once placed, labels can be moved to new positions or hidden while remaining associated with the trace.

Create Label

1. Select **Label** from the context menu, or touch the **Label** Action toolbar button on the trace setup dialog.



2. On the Trace Annotation pop-up, touch **Add Label**.
3. Enter the **Label Text**.
4. Optionally, enter the **Horizontal Pos.** and **Vertical Pos.** (in same units as the trace) at which to place the label. The default position is 0 ns horizontal. **Use Trace Vertical Position** places the label immediately above the trace.

Reposition Label

Drag-and-drop labels to reposition them, or change the position settings on the Trace Annotation pop-up.

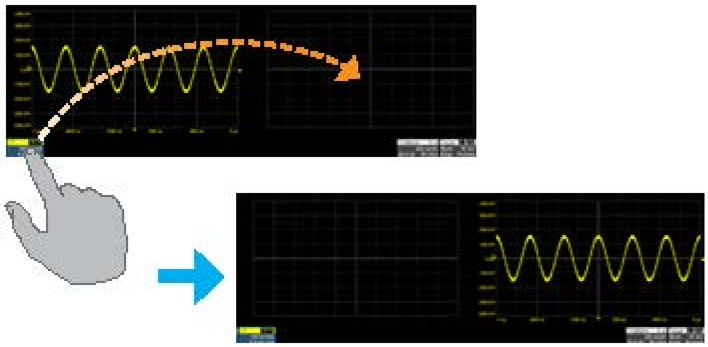
Edit/Remove Label

On the Trace Annotation pop-up, select the **Label** from the list. Change the settings as desired, or touch **Remove Label** to delete it.

Clear **View labels** to hide all labels. They will remain in the list.

Moving Traces

You can move a trace from one grid to another by dragging its descriptor box to the desired grid. This is a convenient way to quickly re-arrange traces on the display.



If you are in Q-Scape display mode, you can drag the descriptor box to a grid on another tab, provided the grid is visible while you are moving the trace. For example, you cannot drag descriptors across tabs in Q-Scape Single mode, because the target grids aren't visible, but you can in Q-Scape Dual or Q-Scape Mosaic modes. This is true even in extended display. For more information, see Display.

See OneTouch Help for ways to pan traces within the same grid.

Naming Traces (Aliases)

A custom name can be added to the mnemonic associated with a trace on its descriptor box, making the oscilloscope user interface more intuitive. This custom name will appear in reports.

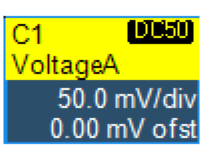
Note: Although there is a 250 character logical limit, we recommend keeping names to 10 characters or less, as characters over this number will be truncated on the display. Custom aliases apply only to the oscilloscope display; use the original trace mnemonic (C1-C*n*, F1-F*n*, etc.) to refer to traces in remote control programs.

Adding Name

1. Right-click (touch-and-hold) on the trace descriptor box to display the context menu, then select **Add/Edit Name**.

   Tip: You can also select Add/Edit Name from the trace Action toolbar.

2. On the virtual keyboard, enter the new name (alias) and click **OK**.



The alias will appear wherever this trace is referenced on the user interface. On trace descriptor boxes, the original trace label will appear above the alias.

Removing Name

To remove an alias from the trace, open the trace context menu and choose **Remove Name**.

Finding Waveform Status

If a small letter "i" inside a bubble appears on the trace descriptor box, this indicates there is additional information regarding a waveform processing error that resulted in the oscilloscope being unable to display a trace, or producing a trace that could lead to unreliable measurements.

To find out the additional information behind this indicator, use the XStreamBrowser installed on the oscilloscope to view the Automation result interface:

app.*Subsystem*.*Wfmn*.Out.Result.Samples

For example, if the F1 math function trace descriptor shows this indicator, look at:

app.Math.F1.Out.Result.Samples

The Status property of Out. Result.Samples will show the value of the error status bit that was generated (there is no value if there is no error). See the *MAUI Oscilloscopes Remote Control and Automation Manual* for instructions on using XStreamBrowser to view Automation results.

Use the table below to look up the status bit meaning.

Bit #	Value	Description
0	0x0000000000001	**Invalid**
1	0x0000000000002	**Overflow**
2	0x0000000000004	**Underflow**
3	0x0000000000008	**Contains Undefined Values**
4	0x0000000000010	**Less Than**
5	0x0000000000020	**Greater Than**
6	0x0000000000040	**Not A Pulse**
7	0x0000000000080	**Not Cyclic**
8	0x0000000000100	**Averaged**
9	0x0000000000200	**Unlocked PLL**
10	0x0000000000400	**Other Error**
11	0x0000000000800	**Other Warning**
12	0x0000000001000	**Other Info**
32	0x0000100000000	**Inputs Incompatible**
33	0x0000200000000	**Algorithm Limits Reached**
34	0x0000400000000	**Bad Definition**
35	0x0000800000000	**Too Few Data**
36	0x0001000000000	**Too Many Data**
37	0x0002000000000	**Uniform HorizInterval Required**
38	0x0004000000000	**Bad Units**
39	0x0008000000000	**Data Range Too Low**

Bit #	Value	Description
40	0x0010000000000	**Data Undersampled**
41	0x0020000000000	**Poor Statistics**
42	0x0040000000000	**Slow Transition Time**
43	0x0080000000000	**Data Resampled**
44	0x0100000000000	**Data Interpolated**
45	0x0200000000000	**Measurement Scale Imprecise**
46	0x0400000000000	**No Data Available**
47	0x0800000000000	**Some Cummulated Results Invalid**
48	0x1000000000000	**Insufficient Memory**
49	0x2000000000000	**Channel Not Active**
50	0x4000000000000	**Use Status Description**

Zooming

Zooms magnify a selected region of a trace by altering the horizontal and/or vertical scale relative to the source trace. Zooms may be created in several ways, by using either the front panel or the touch screen. You can adjust zooms the same as any other trace by using the front panel Vertical and Horizontal knobs or the touch screen zoom factor controls.

The current settings for each zoom trace can be seen on the Z*n* dialogs, while the Zoom dialog is a convenient panel for turning on/off different zooms or changing the zoom source.

The MultiZoom feature locks the Horizontal Scale and Center of those traces that you choose to include in the MultiZoom group. As you change the scale or position of any trace in the group, all traces change in unison. For more information, see MultiZoom.

Zoom Dialog

Zoom controls appear throughout the oscilloscope software wherever it is likely you may wish to change the display scale of a trace—sometimes as a main dialog, sometimes as a subdialog, as when setting up:

- Math traces on F*n* dialogs
- Memory traces on M*n* dialogs

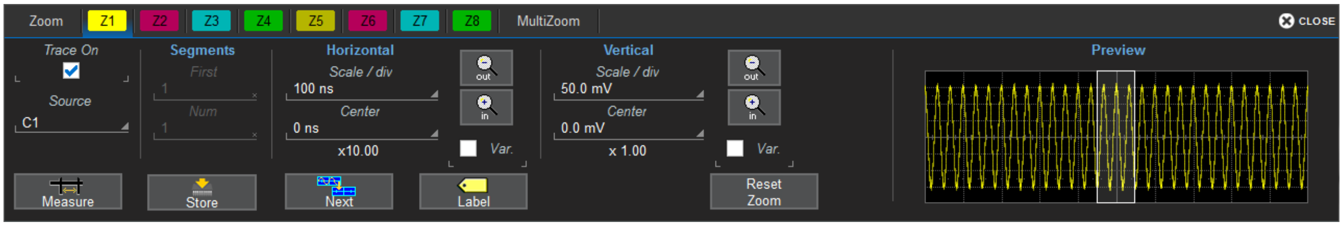
To display the Zoom dialog, choose **Math > Zoom Setup** from the menu bar.

The main Zoom dialog contains selection boxes for turning on/off and changing the source of a zoom. There are also options to:

- **Reset All**, return all zooms to x1 magnification.
- **Quick Zoom**, create a corresponding zoom trace for each open channel trace, same as using the front panel Zoom button.
- Turn on MultiZoom for all active zooms.

Zn Dialog

Behind the main Zoom dialog is a separate tab for each potential zoom trace (Z1-Z*n*). Each Z*n* dialog reflects the center and scale for that zoom. Use it to adjust each zoom independently.



Trace Controls

Trace On shows/hides the zoom trace. It is selected by default when the zoom is created.

Source lets you change the source of the zoom to any digital, math or memory trace while maintaining all other settings.

Segment Controls

These controls are used only in Sequence Sampling Mode.

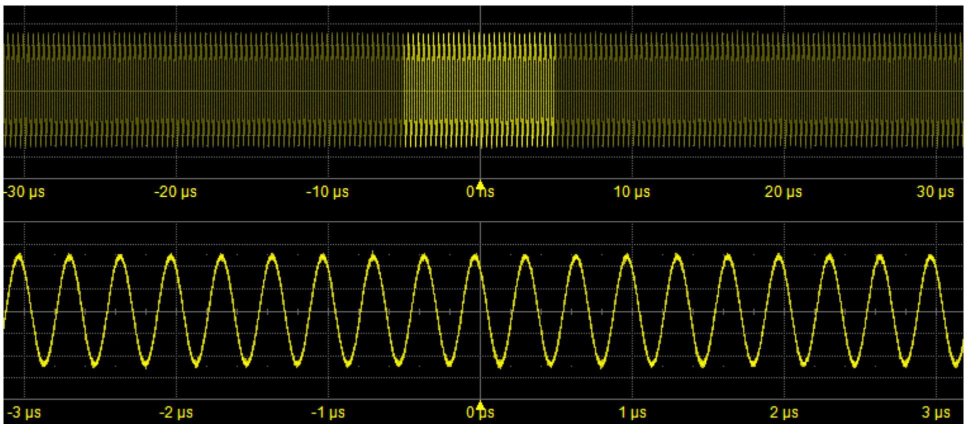
Zoom Factor Controls

- **Out** and **In buttons** increase/decrease zoom magnification and consequently change the Horizontal and Vertical Scale settings. Touch either button until you've achieved the desired level.
- **Var.checkbox** enables zooming in single increments.
- **Horizontal Scale/div** sets the time represented by each horizontal division of the grid. It is the equivalent of Time/div in channel traces, only unlike that setting, it may differ for each zoom trace.
- **Vertical Scale/div** sets the voltage level represented by each vertical division of the grid; it's the equivalent of V/div in channel traces.
- **Horizontal/Vertical Center** sets the time/voltage at the center of the grid.
- **Reset Zoom** returns the zoom to x1 magnification.

Creating Zooms

Any type of trace can be zoomed by creating a new zoom trace (Z*n*) following the procedures here. Zoom traces open in the next empty grid, with the zoomed portion of the source trace highlighted. If there are no more available grids, zooms will open in the same grid as the source trace.

Note: On most instruments with OneTouch, traces can be "zoomed" by pinching/unpinching two fingers over the trace, but this method does not create a separate zoom trace. With channel traces, pinching will alter the acquisition timebase and the scale of all traces. Create a separate zoom trace if you do not wish to do this.



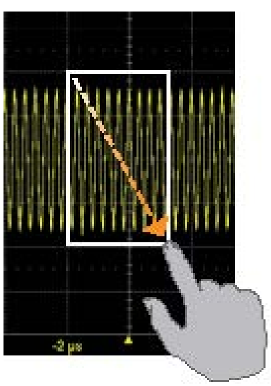
Zoomed area of original trace highlighted. Zoom in new grid below.

Quick Zoom

Use the front panel Zoom button to quickly create one zoom trace for each displayed channel trace. Quick zooms are created at the same vertical scale as the source trace and 10:1 horizontal magnification. The MultiZoom feature is turned on, and all the new zooms are automatically added to the MultiZoom group.

To turn off the quick zooms, press the Zoom button again.

Manual Zoom



To "rectangle zoom", touch-and-drag diagonally to draw a rectangle around the part of the source trace you wish to magnify, then choose a destination: the same trace to rescale the source, or a Z*n* location to create a new zoom trace.

The zoom will expand the horizontal area to fit the full width of the grid, while the vertical area will be rescaled proportionally. The degree of vertical and horizontal magnification, therefore, depends on the size of the rectangle that you draw.

Alternatively, with OneTouch you can drag any Z*n* descriptor box over the Add New box, or touch the Add New box and choose Zoom from the pop-up menu. The next available zoom trace opens with its Z*n* dialog displayed for you to modify scale as needed.

Finally, you can create a Zoom math function. This method creates a new F*n* trace, rather than a new Z*n* trace, but it can be rescaled in the same manner. It is a way to create more zooms than you have Z*n* slots available on your instrument.

Adjust Zoom Scale

The zoom's horizontal units will differ from the signal timebase because the zoom is showing a calculated scale, not a measured level. This allows you to adjust the zoom factor using the front panel knobs or the zoom factor controls however you like without affecting the timebase (a characteristic shared with math and memory traces).

Close Zoom

New zooms are turned on and visible by default. If the display becomes too crowded, you can close a particular zoom and the zoom settings are saved in its Z*n* slot, ready to be turned on again when desired.

To close the zoom, right-click (touch-and-hold until the white box appears) on the zoom descriptor box, then from the context menu choose **Off**.

MultiZoom

The MultiZoom feature unifies the Horizontal Scale (Time/div) setting of those traces that you choose to include in the MultiZoom group. The traces can be zooms or other types of calculated (as opposed to acquired) time traces, such as memories or math functions. This allows you to zoom in/out on several traces in unison, even if they represent different sources, and move the entire group together as one trace is repositioned by dragging or using the Auto Scroll controls.

Note: Traces that cannot be in the same MultiZoom group because they are not time traces or are different record lengths are excluded from selection.

The default MultiZoom sets traces to the **Same Zoom Position** (Horizontal Center). You may optionally choose an **Independent Zoom Position** in order to simultaneously view different parts of the acquisition in different traces, while continuing to zoom and scroll in unison.

MultiZoom is automatically enabled when you use the Quick Zoom method, and all Quick Zooms are added to the MultiZoom group. The descriptor boxes of those traces in the group are marked with a green underline.

When MultiZoom is enabled manually from the Zoom or MultiZoom dialogs, active zooms are automatically added to the MultiZoom group, although other types of calculated traces must be added manually. As you add traces to the group by checking the selection box on the MultiZoom dialog, they are set to the same Horizontal Scale as the *first trace that was added to the group* since MultiZoom was last enabled. Once the group is set, modifying any member of the group using the front panel Horizontal knobs or touch screen Horizontal controls will change the entire group.

Note: Since zooms are added first, the lowest numbered zoom in the group is usually the "master" trace to which all others are rescaled, even though it may not appear at the top of the list of eligible traces on the MultiZoom dialog. When MultiZoom is used with decoded Serial Data waveforms, the result table functionality overrides this behavior, and zooms created by touching a table cell will center on that part of the acquisition.

MultiZoom does not affect the Vertical Scale and Offset of any traces in the MultiZoom group. These continue to be controlled independently for each trace.

Set Up MultiZoom

1. Choose **Math > Zoom Setup...** to open the Zoom dialog.

2. Check the **MultiZoom** box. All active zooms are added to the group and rescaled to match the first active zoom in the list.

3. To modify the group further by adding or removing other types of traces, open the **MultiZoom dialog** and select all the traces that are **In** the MultiZoom group.

4. To change the **Zoom Position** of any trace in the group, select **Independent**, then open the trace dialog and change the **Horizontal Center**.

> Note: Scrolling or dragging any of the MultiZoom group traces will shift the Horizontal Center of this trace by the same amount of Delay as the others, however, it will remain proximate to the center you set, which can be useful when wanting to step through two different sections of a longer acquisition.

Scroll Waveforms

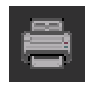
The Auto-Scroll controls appear at the right of the MultiZoom dialog. They work similarly to A/V controls to allow you to continuously scroll all the selected traces in time-locked steps from the beginning to the end of the acquisition.

Print/Screen Capture

The Print function captures an image of the touch screen. What it does with the image next depends on your Print setting:

- Send to a networked printer
- Copy to clipboard to paste into another program
- "Print" to an image file using your Screen Image Preferences
- Email the image file to a preset address

Go to **File > Print Setup** to make the selection on the Print dialog.

Either choosing **File > Print** or touching the **Print Now button**  at the right of the Print dialog will execute your print setting.

The front panel **Print button** will also carry out the selected Print function.

You can also generate an image by choosing **File > Save > Screen image** and touching **Save Now** at the right of the dialog. The file is saved using your latest Screen Image Preferences settings.

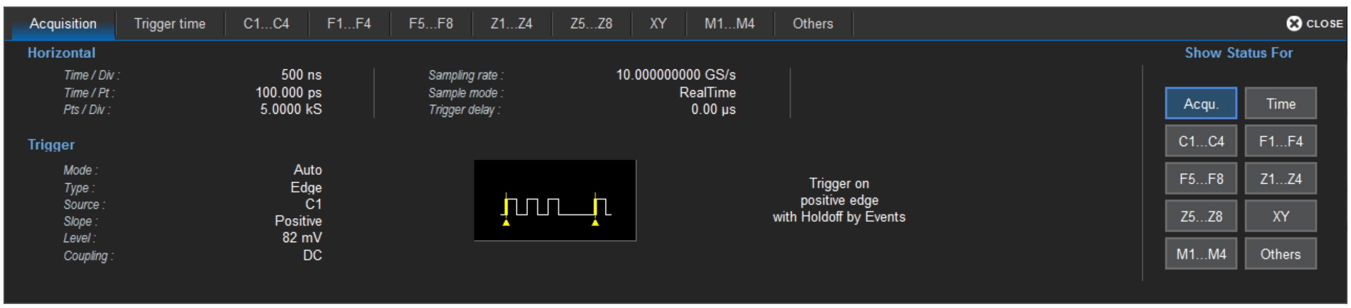
Acquisition

The acquisition settings include everything required to produce a visible trace on screen and an acquisition record that may be saved for later processing and analysis:

- Vertical axis scale at which to show the input signal, and probe characteristics that affect the signal
- Horizontal axis scale at which to represent time, sampling mode and sampling rate
- Acquisition trigger mechanism

Optional acquisition settings include bandwidth filters and pre-processing effects, vertical offset, and horizontal trigger delay, all of which affect the appearance and position of the waveform trace.

All current acquisition settings can be viewed through the various Status dialogs. Access them by choosing the Status option from the Vertical, Timebase or Trigger menus.



Auto Setup

Auto Setup configures the essential acquisition settings based on the first input signal it finds, starting with C1. If nothing is connected to C1, it searches C2 and so forth until it finds a signal. Vertical Scale (Volts/div), Offset, Timebase (Time/div), and Trigger are set to an Edge trigger on the first, non-zero-level amplitude, with the entire waveform visible for at least 10 cycles over 10 horizontal divisions.

To run Auto Setup:

1. Press the front panel **Auto Setup button**, or choose **Auto Setup** from the Vertical, Timebase, or Trigger menus (these all perform the same function).
2. To confirm, press the **Auto Setup** button again, or use the touch screen display.

After running Auto Setup, you'll see the words "Auto Setup" next to an Undo button at the far right of the menu bar. This allows you to restore the settings in place prior to the Auto Setup.

Caution: You will undo all new measurements or math function definitions entered since the Auto Setup when you Undo the Auto Setup.

Vertical

Vertical, also called Channel, settings usually relate to voltage level and control traces along the Y axis.

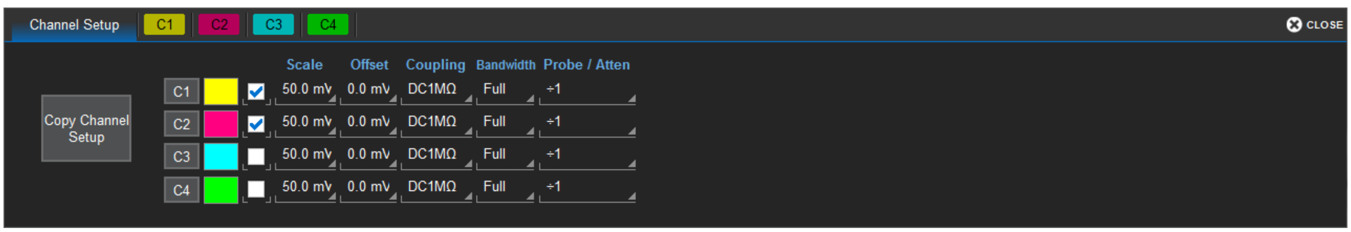
Note: While Digital settings can be accessed through the Vertical menu on Mixed Signal oscilloscopes, they are handled quite differently. See Digital.

The amount of voltage displayed by one vertical division of the grid, or Vertical Scale (V/div), is most quickly adjusted by using the front panel **Vertical knob**. The C*n* descriptor box always shows the current Vertical Scale setting.

Detailed configuration for each trace is done on the C*n* dialogs. Once configured, channel traces can be quickly turned on/off or modified using the Channel Setup dialog.

Channel Setup Dialog

Use the Channel Setup dialog to quickly make basic Vertical settings for all analog input channels. To access the Channel Setup dialog, choose **Vertical > Channel Setup** from the menu bar.



To turn on/off the channel trace, select/deselect the checkbox.

To change the trace color, touch the color block, then choose the new color from the pop-up.

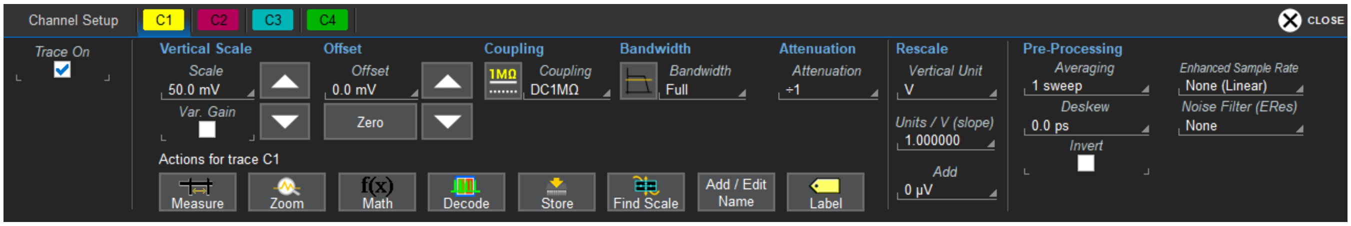
To change any other Vertical settings, touch the input field and enter the new value.

You can also touch **Copy Channel Setup** and select the channels to **Copy From** and **Copy To**.

Tip: On instruments with OneTouch, you can copy settings from one channel to another just by dragging the source channel descriptor box onto the target channel descriptor box.

C*n* (Channel) Dialog

Full vertical setup is done on the C*n* dialog. To access it, choose **Vertical > Channel*n* Setup** from the menu bar, or touch the **Channel descriptor box**.



If a Teledyne LeCroy probe is connected, its Probe dialog appears to the right of the C*n* dialog.

Note: In case of a waveform processing error (e.g., overflow), a small letter "i" inside a bubble will appear on the F*n* descriptor box to indicate there is information regarding the waveform status. See Finding Waveform Status for instructions on finding the error.

Vertical Settings

The **Trace On** checkbox turns on/off the channel trace.

Vertical Scale sets the gain (sensitivity) in the selected Vertical units, Volts by default. Select **Variable Gain** for fine adjustment or leave the checkbox clear for fixed 1, 2, 5, 10-step adjustments.

Offset adds a defined value of DC offset to the signal as acquired by the input channel. This may be helpful in order to display a signal on the grid while maximizing the vertical height (gain) of the signal. A negative value of offset will "subtract" a DC voltage value from the acquired signal (and move the trace down on the grid) whereas a positive value will do the opposite. Touch **Zero Offset** to return to zero.

A variety of **Bandwidth** filters are available. To limit bandwidth, select a filter from this field.

Coupling may be set to DC 50 Ω, DC1M, AC1M or GROUND.

Caution: The maximum input voltage depends on the input used. Limits are displayed on the body of the instrument. Whenever the voltage exceeds this limit, the coupling mode automatically switches to GROUND. You then have to manually reset the coupling to its previous state. While the unit does provide this protection, damage can still occur if extreme voltages are applied.

Probe Attenuation

Probe **Attenuation** values for third-party probes may be entered manually on the C*n* dialog. The instrument will detect it is a third-party probe and display these fields.

When a Teledyne LeCroy probe is connected to a channel input, the Attenuation field becomes a button to access the Probe dialog, a tab added to the right of the C*n* tab. Enter Attenuation on the Probe dialog.

Rescale Settings

The rescale settings provide the same capability as the oscilloscope Rescale math function (y=*a*x + *b*, where the original value is x, Units/V is *a*, and Add is *b*), only applied directly to the input trace rather than to a separate math function trace.

Enter the number of units equal to 1 Volt in **Units/V** and any additive constant in **Add**.

The **Vertical Units** setting may be changed from Volts (V) to Amperes (A) or Other units of measure. This is useful when using a third-party current probe (which is not auto-detected) or when probing across a current sensor/resistor. Units assigned directly to an input will carry to other traces calculated using that input, such as math or spectrum traces (FFT).

When selecting to use a unit other unit than volts or amperes, first choose the unit subcategory from the pop-up dialog that appears, then the unit.

Pre-Processing Settings

Pre-processing functions modify the acquired signal prior to display, math and measurement processing.

Average performs continuous averaging, the repeated addition, with unequal weight, of successive source waveforms. It is particularly useful for reducing noise on signals drifting very slowly in time or amplitude. The most recently acquired waveform has more weight than all the previously acquired ones: the continuous average is dominated by the statistical fluctuations of the most recently acquired waveform. The weight of old waveforms in the continuous average gradually tends to zero (following an exponential rule) at a rate that decreases as the weight increases.

Interpolate applies (Sinx)/x interpolation to the waveform. The selection of None or Linear applies Linear interpolation, which inserts a straight line between sample points and is best used to reconstruct straight-edged signals such as square waves. (Sinx)/x interpolation, on the other hand, is suitable for reconstructing curved or irregular wave shapes, especially when the sample rate is 3 to 5 times the system bandwidth. Choose an upsample factor of 2 or more points. The Interpolation setting is disabled .

On HDO/MDA "A" models, this setting is called **Enhanced Sample Rate** and appears disabled when using a sample rate greater than 2.5 GS/s, as the system automatically sets the upsample factor according to your sample rate. Only when the sample rate is below this can you choose an upsample factor or use Linear interpolation (None).

Deskew adjusts the horizontal time offset by the amount entered in order to compensate for propagation delays caused by different probes or cable lengths. The valid range is dependent on the current timebase setting. The Deskew pre-processing setting and the Deskew math function perform the same action.

Noise Filter applies Enhanced Resolution (ERes) filtering to increase vertical resolution, allowing you to distinguish closely spaced voltage levels. The tradeoff is reduced bandwidth. ERes functions similarly to smoothing the signal with a simple, moving-average filter. It is best used on single-shot acquisitions, acqusitions where the data record is slowly repetitive (and you cannot use averaging), or to reduce noise when your signal is noticeably noisy but you do not need to perform noise measurements. It also may be used when performing high-precision voltage measurements and zooming with high vertical gain, for example. ERes is disabled.

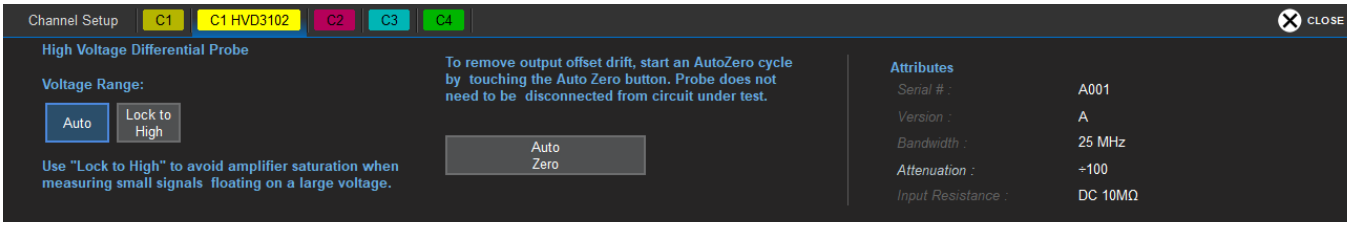
Invert changes the apparent polarity of the signal, substituting an equivalent negative value for a positive one, and vice versa, so that the waveform appears to be "flipped" on screen.

Cable De-Embedding allows you to remove the effects of cables used in the test setup. Select the checkbox, then enter the attenuation constants or cable characteristics (provided by the cable manufacturer) on the Cable De-Embedding dialog that appears. See Cable De-Embedding.

Probe Dialog

The Probe Dialog immediately to the right of the C*n* dialog displays the probe attributes and (depending on the probe type) allows you to control the probe from the touch screen.

Caution: Remove probes from the circuit under test before initializing Auto Zero or DeGauss.



Depending on the type of probe you have connected to the channel, you will see any of the following controls:

Power On initiates power to active probes via the oscilloscope interface.

LED Active turns on AutoColor ID if the probe has this feature. The LED on the probe body will light in the color of the channel to which the probe is connected.

Auto Zero corrects for DC offset drifts that naturally occur from thermal effects in the amplifier of active probes. Teledyne LeCroy probes incorporate Auto Zero capability to remove the DC offset from the probe's amplifier output to improve the measurement accuracy.

The **Degauss** control is activated for some types of probes (e.g., current probes). Degaussing eliminates residual magnetization from the probe core caused by external magnetic fields or by excessive input. It is recommended to always Degauss probes prior to taking a measurement.

If using a WaveLink probe, touch **Tip Select** and select the type of tip/lead you are using.

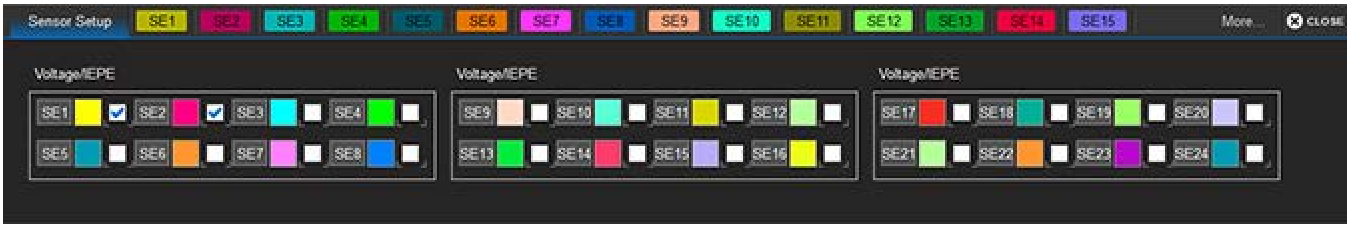
Note: Making the tip selection here results in the amplifier and tip combination having the response calibrated for at the factory. Failure to do so may result in inaccurate measurements.

On oscilloscopes running MAUI version 8.5.1.1 or later, HVD3000 probes set attenuation relative to the oscilloscope’s V/div setting and the **Voltage Range** selection:

- **Auto** automatically raises attenuation when V/div is >7.9 or lowers attenuation when V/div is <7.9, allowing you to properly view the input waveform.
- **Lock to High** locks attenuation to the highest setting, regardless of the V/div setting. Maintaining a high attenuation will allow small signals on larger voltage waveforms to be accurately measured.

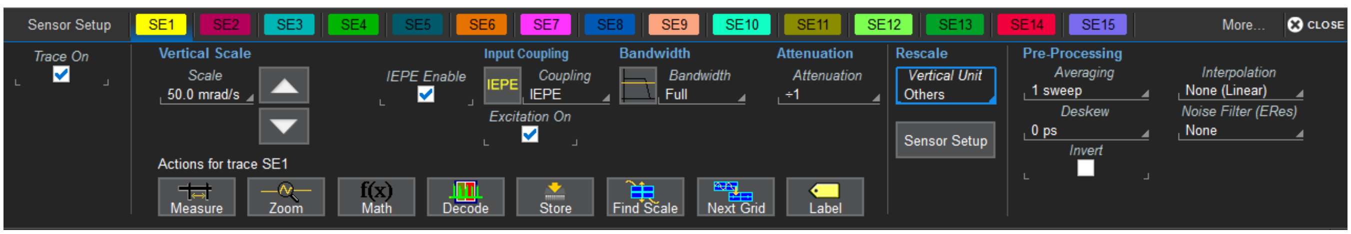
Sensor Setup

If your system includes a SAM40 Sensor Acquisition Module (available for High Definition oscilloscopes), the Vertical menu will offer an option for PMU Sensor Menu. Choose this to open the Sensor Setup dialog.



Similar to the Channel Setup dialog, Sensor Setup is a collection of all the available sensor inputs, allowing you to quickly enable/disable an input. Behind it are the individual Sensor (SE*n*) configuration dialogs.

SE*n* (Sensor) Dialog



The Sensor dialogs contain many of the same settings as the Channel (C*n*) configuration dialogs, and many function in a similar manner. However, sensor configuration differs from analog channel configuration in the following respects.

As with channel traces, the **Vertical Scale** sets the amount of voltage represented by one Vertical division of the grid.

If using an IEPE/ICP compatible sensor (IEPE mode), always make the selection IEPE Sensor in the **Coupling** field. Other types of coupling may be selected if using a BNC cable for other voltage input to the SAM40 (Voltage mode).

When the **IEPE Enable** checkbox is selected, the SAM40 applies an excitation voltage to ICP/IEPE-compatible sensors.

The available **Bandwidth** settings are changed when inputting a sensor signal, generally to lower values than are available for oscilloscope channels. Unlike the hardware bandwidth filters on the oscilloscope channel inputs, SAM40 inputs utilize a digital FIR filter to limit bandwidth.

Most sensors output only voltage values. The SAM40 **Rescale** settings allow you to change the units in which sensor output values are displayed to a much wider selection than Volts or Amperes. They can also be used to apply intelligent rescaling of output values, equivalent to using the Rescale math function (y=ax+b), where a is the multiplication factor (Units/V) and b is the additive constant (Add).

Note: Channel inputs can be rescaled on the C*n* dialog, while the output unit of measurement parameters and math functions can be changed using the Units subdialog that appears next to the respective P*n* or F*n* dialog. The same units of measure are supported.

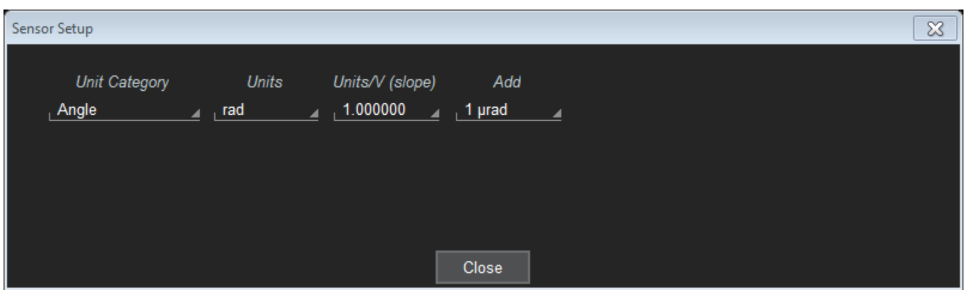
As with channel rescaling, measurements performed on this sensor are converted and displayed in the selected unit. However, math functions will be displayed in the unit that is logically appropriate for the result, including complex results that must be derived from a table of core SI units.

Example: Multiplying one sensor trace in Volts per meter by another sensor trace in meters yields a result in Volts.

Units are automatically rescaled up or down within the list of standard, SI prefixes (micro, milli, centi, kilo, etc.) based on the relative size of the sensor signal.

Example: A 1000 V reading is shown as 1 kV, while a .1 V reading is shown as 100 mV. When the multiplication factor is 1 V = Pascals (Pa), a 1 mV reading is displayed as 1 mPa rather than .001 Pa or 100e-3 Pa.

To rescale a sensor trace to units other than Volts or Amperes, choose **Vertical Unit Others**, then select the **Sensor Setup** button. This will display the following pop-up:



Rescaling from the Sensor dialog.

Unit Category is the unit type, for example, Length or Velocity.

Units reflect the Category selection, for example, a Length unit of meters (m) versus a Velocity unit of meters per second (m/s).

Units/V (slope) is the multiplication factor (*a*) to use to scale the acquired sensor Voltage. Enter the number of the new unit that will equal 1 Volt.

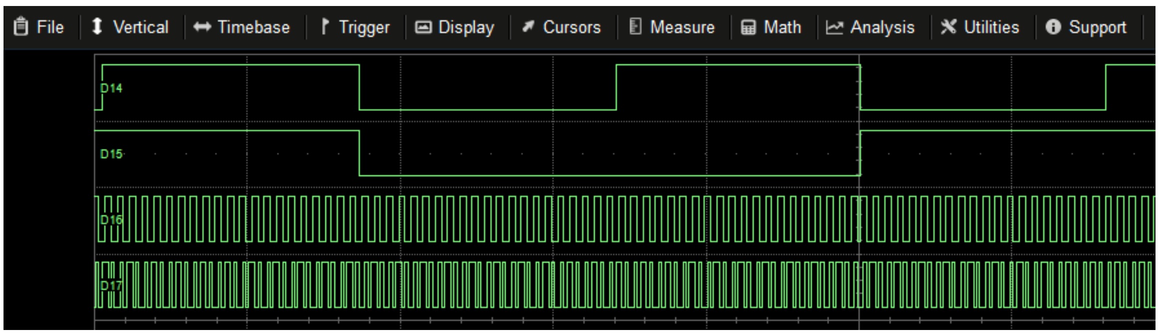
Add is the amount of additive constant (*b*) to add to the rescaled value. This value is always given in the new unit.

Digital (Mixed Signal)

When a Mixed Signal device is connected to the oscilloscope, digital input options are added to the Vertical menu. There are set up dialogs for each possible digital group, Digital1 to Digital*n*, which correspond to digital buses. You choose which lines make up each digital group, what they are named, and how they appear on the display.

Digital Traces

When a digital group is enabled, digital Line traces show which lines are high, low, or transitioning relative to the threshold. You can also view a digital Bus trace that collapses all the lines in a group into their Hex values.



Four digital lines displayed with a Vertical Position +4.0 (top of grid) and a Group Height 4.0 divisions. Depending on your input method, Height may be defined by the entire group or by the individual line.

Activity Indicators

Activity indicators appear at the bottom of the Digital*n* dialogs. They show which lines are High (up arrow), Low (down arrow), or Transitioning (up and down arrows) relative to the Logic Threshold value, providing a quick view of which lines are of interest to display on screen.

Digital Setup Using Digital Leadset

The digital leadset enables input of up-to-16 lines of digital data. Physical lines can be preconfigured into different logical groups, Digital*n*, corresponding to a bus. The transitions for each line may be viewed through different displays.

The digital leadset features two digital banks with separate Threshold and Hysteresis controls, making it possible to simultaneously view data from different logic families.

Initially, logical lines are named and numbered the same as the physical lead they represent, although any line can be renamed appropriately or re-assigned to any lead.

Connecting/Disconnecting the Leadset

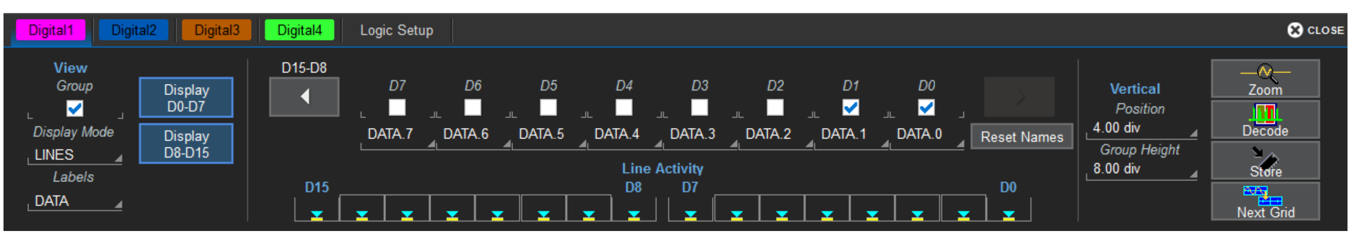
To connect the leadset to the instrument, push the connector into the Mixed Signal interface below the front panel until you hear a click.

To remove the leadset, press and hold the buttons on each side of the connector, then pull out to release.

Each flying lead has a signal and a ground connection. A variety of ground extenders and flying ground leads are available for different probing needs.

To achieve optimal signal integrity, connect the ground at the tip of the flying lead for *each* input used in your measurements. Use either the provided ground extenders or ground flying leads to make the ground connection.

Digital Group Setup



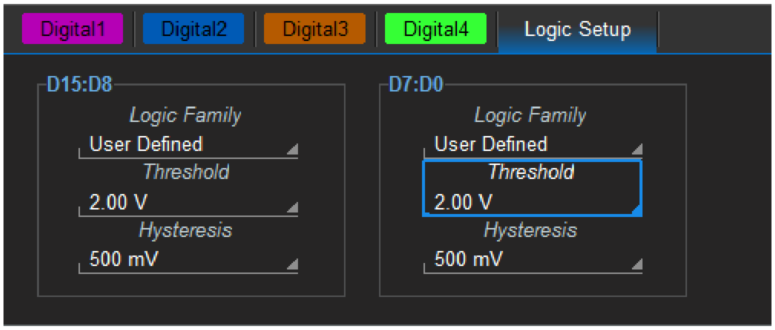
To set up a digital group:

1. From the menu bar, choose **Vertical > Digital*n* Setup**.
2. On the Digital*n* set up dialog, check the boxes for all the lines that comprise the group. Touch the Right and Left Arrow buttons to switch between digital banks as you make line selections.

Note: Groups can include from one to all of the leads from any digital bank.

3. Check **View Group** to start the display.

4. When you're finished on the Digital*n* dialog, open **Logic Setup** and choose the **Logic Family** that applies to each digital bank, or set custom **Threshhold** and **Hysteresis** values.



Digital Display Setup

Choose the type and position of the digital traces that appear on screen for each digital group.

1. Choose a **Display Mode**:

   - **Lines** (default) shows a time-correlated trace indicating high, low, and transitioning points (relative to the Threshold) for *every* digital line in the group. The size and placement of the lines depend on the number of lines, the Vertical Position and Group Height settings.

   - **Bus** collapses the lines in a group into their Hex values. It appears immediately below all the Line traces when both are selected.

   - **Line & Bus** displays both types of digital trace.

2. In **Vertical Position**, enter the number of divisions (positive or negative) relative to the zero line of the grid where the display begins.The top of the first trace appears at this position.

3. In **Group Height**, enter the total number of grid divisions the entire display should occupy. All the selected traces (Line and Bus) will appear in this much space. Individual traces are resized to fit the total number of divisions available.

To close digital traces, uncheck the **Group box** on the Digital*n* dialog.

Tip: Because a new grid opens to accommodate each enabled group, you may wish to enable groups one or two at a time when they have many lines to maximize the total amount of screen space available for each. Closing the set up dialogs will also increase available screen space.

Renaming Digital Lines

The labels used to name each line can be changed to make the user interface more intuitive.

Touch **Label** and select the type:

- **Data** (default) appends "D." to the front of each line number.
- **Address** appends "A." to the front of each line number.
- **Custom** lets you create your own labels line by line. If using Custom labels:
  - > Touch the **Line number field** below the corresponding checkbox. If necessary, use the **Left/Right Arrow buttons** to switch between banks.
  - > Use the virtual keyboard to enter the name, then press **OK**.

Renumbering Digital Lines

Labels can also be "swapped" between lines. This procedure helps in cases where the physical lead number is different from the logical line number you would like to assign to that input. It can save time having to reattach leads or reconfigure groups.

Example: A group is set up for lines 0-4, but lead 5 was accidentally attached to the probing point. By "swapping" line 5 with line 4, you do not need to change either the physical or the logical setup.

1. Select a **Label** of **Data** or **Address**.
2. Touch the **Line number field** below the corresponding checkbox. If necessary, use the **Left/Right Arrow buttons** to switch between banks.
3. From the pop-up, choose the line with which you want to swap labels.

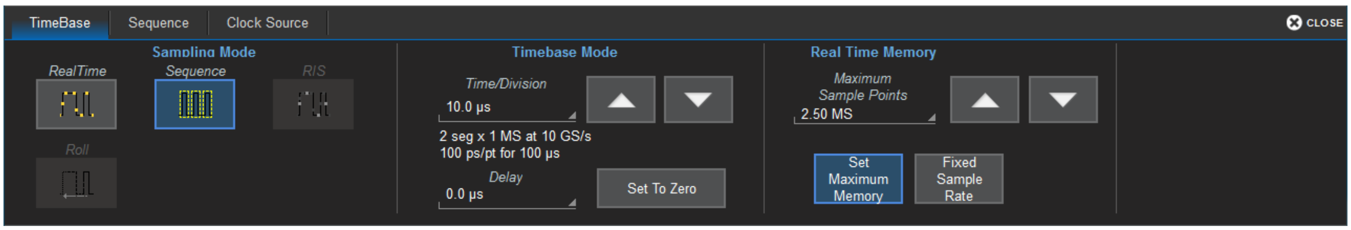
Timebase

Timebase (Horizontal) settings control traces along the X axis. The timebase is shared by all channels.

The time represented by each horizontal division of the grid, or **Time/Division**, is most easily adjusted using the **front panel Horizontal knob**. Full Timebase set up is done on the Timebase dialog, accessed either by choosing **Timebase > Horizontal Setup** from the menu bar or by touching the **Timebase descriptor box**.

Timebase Set Up

Use the Timebase dialog to select the Sampling Mode, and Memory /Sample Rate. You can also use it instead of the Front Panel to modify the Time/Div and horizontal Delay.



Sampling Mode

The **Sampling Mode** determines how the instrument samples the input signal and renders it for display. See Sampling Modes for a description of each type.

Timebase Mode

Time/Division is the time represented by one horizontal division of the grid. Touch the Up/Down Arrow buttons on the Timebase dialog or turn the front panel Horizontal knob to adjust this value. The overall length of the acquisition record is equal to 10 times the Time/Division setting.

Delay is the amount of time relative to the trigger event to display on the grid. Raising/lowering the Delay value has the effect of shifting the trace to the right/left. This allows you to isolate and display a time/event of interest that occurs before or after the trigger event.

- **Pre-trigger Delay**, entered as a negative value, displays the acquisition time prior to the trigger event, which occurs at time 0 when in Real Time sampling mode. Pre-trigger Delay can be set up to the instrument's maximum sample record length; how much actual time this represents depends on the timebase. At maximum pre-trigger Delay, the trigger point is off the grid (indicated by the arrow at the lower right corner), and everything you see represents 10 divisions of pre-trigger time.
- **Post-trigger Delay**, entered as a positive value, displays time following the trigger event. Post-trigger Delay can cover a much greater lapse of acquisition time than pre-trigger Delay, up to the equivalent of 10,000 divisions after the trigger event (it is limited at slower time/div settings and in Roll mode). At maximum post-trigger Delay, the trigger point is off the grid far left of the time displayed.

Set to Zero returns Delay to zero.

Sampling Rate

These controls specify how the instrument samples when in Real Time mode.

Maximum Sample Points is the number of samples taken per acquisition when you choose to Set Maximum Memory. It changes to show **Sampling Rate**, the number of samples taken per time division, when using a Fixed Sampling Rate.

Set Maximum Memory, the default setting on MAUI oscilloscopes, automatically adjusts the sampling rate to take the maximum number of samples possible given the amount of pre- or post-trigger Delay and the Time/div, up to the maximum record length. This optimizes the sample rate for fast timebases when in Real Time mode.

Fixed Sampling Rate activates the Sampling Rate field for you to set your own rate. The oscilloscope will use whatever Memory as required for the acquisition time, until none remains. Lowering the rate can extend the acquisition to accommodate slower timebases or longer delays.

Note: To avoid aliasing and other waveform distortions, it is advisable (per Nyquist) to acquire at a sample rate at least twice the bandwidth of the input signal. A sample rate 10 times the acquisition bandwidth may be preferable in order to over sample fast signal rise times consistent with the oscilloscope bandwidth. Those with higher speed acquisition systems may need to manually adjust Bandwidth, Time/div, Sample Rate, and the number of Active Channels to achieve the optimal result.

On HDO "A" models, if you choose a rate greater than 2.5 GS/s, the system will automatically enter Enhanced Sample Rate, applying Sinx/x interpolation to prevent aliasing at the higher sample rate. An upsample factor of 2 pts. is used for 5 GS/s timebases, or 4 pts. for 10 GS/s and higher timebases. At lower rates, you can set the Enhanced Sample Rate factor yourself on the C*n* dialog, or choose to use Linear interpolation.

Sensor Sample Rate

If your system includes a SAM40 Sensor Acquisition Module, the far right of the Timebase dialog shows the rate at which the SAM40 is sampling. When the oscilloscope is at a fixed sample rate, the SAM40 will sample at a maximum rate of ≤100 KS/s so long as the oscilloscope Sample Rate and Time/div settings allow it. Increasing Time/div or switching to Maximum Memory will decrease the SAM40 sample rate (as it would the oscilloscope sample rate). Adjusting these settings to yield 100 KS/s or lower sampling on the oscilloscope will cause the sensor traces to track exactly with the channel traces.

Regardless of the SAM40 sample rate, the timebase is locked to the oscilloscope, and sensor traces will always be time synchronous with channel traces.

Sampling Modes

The Sampling Mode determines how the instrument samples the input signal and renders it for display.

Real Time Sampling Mode

Real Time sampling mode is a series of digitized voltage values sampled on the input signal at a uniform rate. These samples are displayed as a series of measured data values associated with a single trigger event. By default (with no Delay), the waveform is positioned so that the trigger event is time 0 on the grid. The relationship between sample rate, memory, and time can be expressed as:

Capture Interval = 1/(Sample Rate x Memory)
Capture Interval/10 = Time Per Division

Usually, on fast timebase settings, the maximum sample rate is used when in Real Time mode. For slower timebase settings, the sample rate is decreased so that the maximum number of data samples is maintained over time.

RIS Sampling Mode

RIS (Random Interleaved Sampling) allows effective sampling rates higher than the maximum single-shot sampling rate. It is available on timebases ≤ 10 ns/div.

The maximum effective RIS sampling rate is achieved by making multiple single-shot acquisitions at maximum real-time sample rate. The bins thus acquired are positioned approximately 8 ps (125 GS/s) apart. The process of acquiring these bins and satisfying the time constraint is a random one. The relative time between ADC sampling instants and the event trigger provides the necessary variation.

Because the instrument requires multiple triggers to complete an acquisition, RIS is best used on repetitive waveforms with a stable trigger. The number depends on the sample rate: the higher the sample rate, the more triggers are required. It then interleaves these segments to provide a waveform covering a time interval that is a multiple of the maximum single-shot sampling rate. However, the real-time interval over which the instrument collects the waveform data is much longer, and depends on the trigger rate and the amount of interleaving required.

Note: RIS is not available when the oscilloscope is operating at fixed sample rate or using another form of digital interleaving (e.g., HD, 2-channel or DBI mode).

Roll Sampling Mode

Roll mode displays incoming points of slow timebase acquisitions so that the trace appears to "roll" continuously across the screen from right to left. The acquisition is complete when a trigger event is detected, at which point the next acquisition begins immediately. Parameters or math functions are updated every time the roll mode buffer is updated as new data becomes available. This resets statistics on every step of Roll mode that is valid because of new data.

Timebase must be set sufficiently slow to enable Roll mode selection; increase Time/div to 100 ms/div or more and increase sample rate to 5 MS/s or less to activate the Roll mode option on the Timebase dialog. Only Edge trigger is supported for Roll mode acquisitions.

Note: Roll mode sampling is not available when using another form of digital interleaving. If processing time is greater than acquisition time, the roll mode buffer is overwritten. The instrument warns, "Channel data is not continuous in ROLL mode!!!" and rolling starts again.

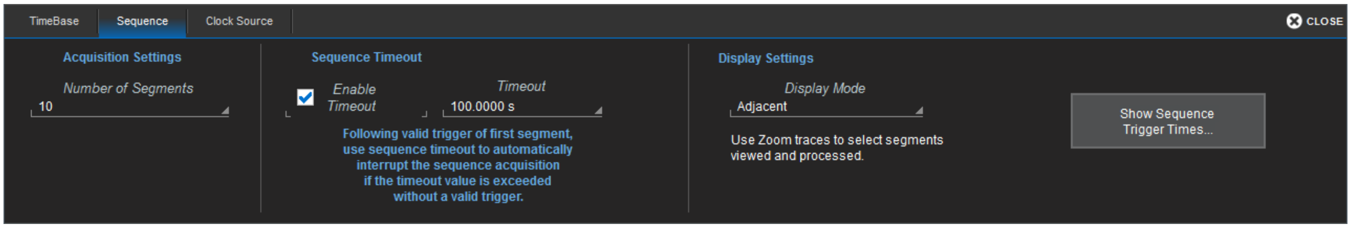
Sequence Sampling Mode

In Sequence Mode sampling, the completed acquisition consists of a number of fixed-size segments each containing the trigger event. The instrument calculates the capture duration and number of sample points in each segment from the user-defined number of segments and total available memory. Acquired segments can be arranged adjacent to one another, forming the waveform display of a typical acquisition, or in other types of sequence displays to facilitate comparison of the segments.

Sequence Mode is ideal for capturing specific events that may be separated by long time intervals. The instrument can acquire over long periods waiting for the trigger event, recording only the desired segments while ignoring the uninteresting periods between events. Measurements can be made on selected segments or on the entire acquisition sequence.

Sequence Mode Set Up

The Sequence dialog appears only when Sequence Mode sampling is selected. Use it to define the number of fixed-size segments to be acquired.

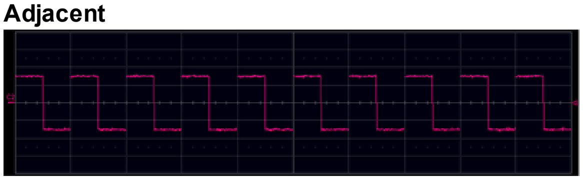


1. From the menu bar, choose Timebase > Horizontal Setup..., then Sequence Sampling Mode.
2. On the Sequence tab under Acquisition Settings, enter the Number of Segments to acquire.
3. To stop acquisition in case no valid trigger event occurs within a certain timeframe, check the Enable Timeout box and provide a Timeout value.

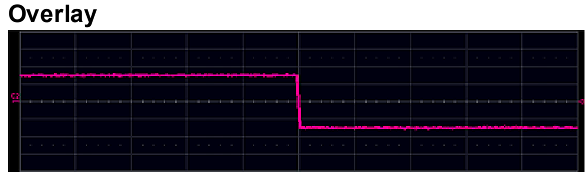
   Note: While optional, Timeout ensures that the acquisition completes in a reasonable amount of time and control is returned to the operator/controller without having to manually stop the acquisition, making it especially useful for remote control applications.

4. Touch Display mode and select a sequence display mode from the pop-up menu:

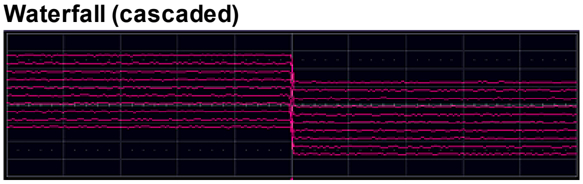
Adjacent



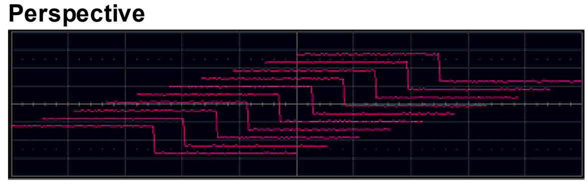
Overlay



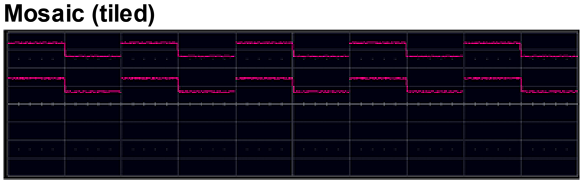
Waterfall (cascaded)



Perspective



Mosaic (tiled)



5. To see the trigger times of those segments acquired, stop acquisition and touch Show Sequence Trigger Times. This will launch the Trigger Time tab of the Acquisition Status dialogs.

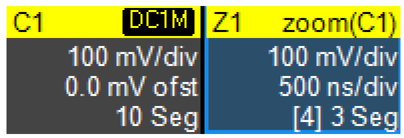
Viewing Sequence Segments

When in Sequence sampling mode, you can view individual segments easily using the front panel **Zoom button**. A new zoom of the channel trace defaults to Segment 1.

You can view other segments by changing the **First** and total **Num**(ber) of segments to be shown on the Z*n* dialog. Touch the Z*n* descriptor box to display the dialog.

Tip: By setting the Num to 1, you can use the front panel Adjust knob to scroll through each segment in order.

Channel descriptor boxes indicate the total number of segments acquired in sequence mode. Zoom descriptor boxes show the first segment displayed and total number of segments displayed ([#] #). As with all other zoom traces, the zoomed segments are highlighted on the source trace.



Example: You have acquired 10 segments. You choose to display segments 4 to 6—or, a total of 3 segments beginning with segment 4. The C*n* descriptor box reads 10. The Z*n* descriptor box reads [4]3 Seg, meaning you are displaying a total of 3 segments, starting with segment 4.

View Segment Time Stamps

To view time stamps for each segment:

1. From the Sequence dialog, choose **Show Sequence Trigger Times**.

   Or

   From the menu bar, choose **Timebase > Acquisition Status**, then open **Trigger time** .

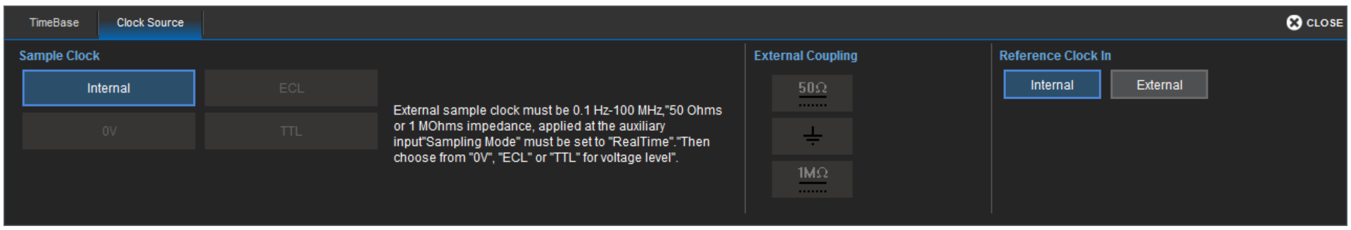
2. Under **Show Status For**, choose **Time**.

Clock Source Settings

An external reference clock (applied via the REF In input) is used to synchronize the instrument's internal timebase to an external frequency source. This allows multiple instruments to lock their timebases to a common source.

An external sampling clock (applied via the EXT input) replaces the internal timebase as the sampling clock. This means that the external sampling clock controls when the digitizers sample the input waveforms.

Note: An external trigger cannot be used when the external sampling clock is in use.



Reference Clock

The default setting is to use the instrument's **Internal** 10 MHz clock. To use an external reference clock:

1. Connect a clock source to the reference clock input using a BNC cable.
2. Go to **Timebase > Horizontal Setup** and choose **Real-Time Sampling Mode**.
3. On the **Clock Source tab** under Reference Clock choose **External**.

Sample Clock

The default setting is to use the instrument's **Internal** clock. To use an external sample clock:

1. Connect a clock source to the sample clock input using a BNC cable.
2. Go to **Timebase > Horizontal Setup** and choose **Real-time Sampling Mode**.
3. On the **Clock Source tab** under **Sample Clock** choose from 0V, ECL, or TTL pulse types.
4. Choose an **External Coupling** that matches the clock source input impedance.

Clock Frequencies

The signal types, frequency ranges, and levels accepted for an external clock are model dependent. See the product datasheet at teledynelecroy.com.

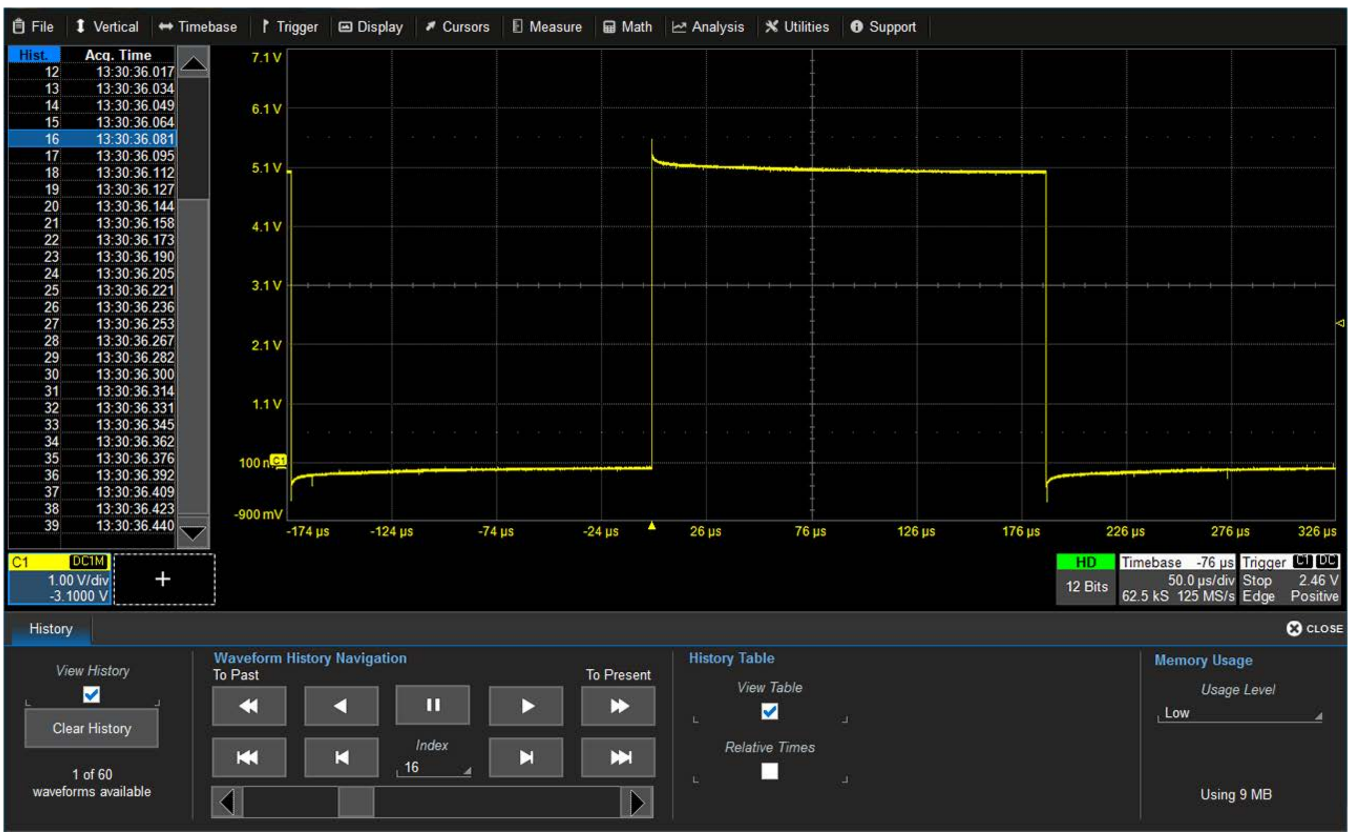
History Mode

History Mode allows you to review any acquisition saved in the history buffer, which automatically stores all acquisition records until full. Not only can individual acquisitions be restored to the grid, you can "scroll" backward and forward through the history at varying speeds to capture changes in the waveforms over time. To access this feature, choose **Timebase > History Mode**, or press the front panel **History Mode button**.

Each record is indexed and time-stamped, and you can choose to view the absolute time of acquisition or the time relative to when you entered History Mode. In the latter case, the last acquisition is time zero, and all others are stamped with a negative time. The maximum number of records stored depends on your acquisition settings and the total available memory.

Entering History Mode automatically stops new acquisitions. To leave History Mode, press the **History Mode button** again, or restart acquisition by pressing one of the front panel Trigger Mode buttons.

> **Note:** History Mode does not work with Sequence Mode acquisitions, Interpolation set on the input channel, or any type of channel interleaving, including RIS sampling mode.



Oscilloscope in History mode.

Replay Acquisition History

This is a good way to begin using History Mode. Watching a "movie" of the history allows you to see waveform changes that are invisible during real-time acquisition. Select **View History** to enable the display, then use the buttons or the slider bar at the bottom of the dialog to navigate the history of acquisitions.

- Top row buttons scroll: Fast Backward, Slow Backward, Slow Forward, Fast Forward.
- Bottom row buttons step: Back to Start, Back One, Go to Index (row #), Forward One, Forward to End.

Press **Pause** when you see something of interest, then use the History table to find the exact Index.

Select Single Acquisition

1. Select **View History** to enable the display, and **View Table** to show the index of records.
2. Optionally, select to show **Relative Times** on the table.
3. To view individual acquisitions, select the row from the table or enter its **Index** number on the dialog.

Trigger

Triggers define the event around which digitized information is displayed on the grid.

Different Trigger Types are used to select different events in the trigger source waveforms: edge voltages, pulse widths, high/low states, etc. These may be a single channel event or a complex pattern of events across several channels. On instruments with Mixed Signal capabilities, pattern triggers can be set on analog channels (including the External Trigger input), digital lines or a mix of both.

In addition to the type, the Trigger Mode determines how the instrument behaves as it encounters trigger events: take a single acquisition and stop, holding on to the display of the last acquisition, or continuously take and display acquisitions.

In both cases, when the previous acquisition has completed processing, the oscilloscope is again ready to acquire and the READY indicator is lit. If, while READY, the trigger circuit detects a signal that matches the trigger conditions, the oscilloscope triggers on the next matching event, and the TRIG'D indicator is lit.

Unless modified by a pre- or post-trigger Delay, the trigger event appears at time 0 at the horizontal center of the grid, and a period of time equal to five divisions of the timebase is shown to the left and right of it. Delay shifts the acquisition "window" on screen, displaying a different portion of the waveform.

An additional condition of Holdoff by time or events is available for Edge and Pattern triggers, including those that appear within Qualified triggers. Holdoff arms the trigger on the first matching event, inserts the holdoff count, then triggers on a subsequent event. Often, especially with repetitive signals, the initial arming event appears to the left of the trigger in "negative" acquisition time.

Trigger Modes

The Trigger Mode determines how often the instrument acquires. It is equivalent to how analog oscilloscopes "sweep," or refresh, the display. Trigger Mode can be set from the Trigger menu or from the front panel Trigger control group.

In **Single** mode, when you choose **Trigger > Single** or press the front panel **Single** button, the oscilloscope readies, arms, and triggers provided all trigger conditions (including Holdoff) are met. It then stops and continues to display the last acquisition until a new one is taken. The oscilloscope remains armed unless manually stopped or triggered, and if a valid trigger does not occur, invoking Single a second time will force a trigger and display the acquisition.

In **Normal** mode, operation is the same as in Single, except that the trigger automatically re-arms after the previous acquisition is complete, and data is continuously refreshed on the touch screen.

Auto operates the same as Normal mode, except that a trigger is forced if the trigger event has not occurred within a preset timeout period.

Stop ceases acquisition processing until you select one of the other three modes. The arming and Holdoff counters are cleared, even if there has not yet been a trigger since the previous acquisition.

Trigger Types

The Trigger Type sets the triggering conditions.

Basic Types

Edge triggers upon a achieving a certain voltage level in the positive or negative slope of the waveform.

Width triggers upon finding a positive- or negative-going pulse width when measured at the specified voltage level.

Pattern triggers upon a user-defined pattern of concurrent high and low voltage levels on selected inputs. In Mixed-Signal oscilloscopes, it may be a digital logic pattern relative to voltage levels on analog channels, or just a digital logic pattern omitting any analog inputs. Likewise, if your oscilloscope does not have Mixed-Signal capability, the pattern can be set using analog channels alone.

TV triggers on a specified line and field in standard (PAL, SECAM, NTSC, HDTV) or custom composite video signals.

Measurement is not technically a trigger, per se, but permits the instrument to stop and display acquisition based on a post-processing measurement result, similar to a trigger.

Serial triggers on the occurrence of user-defined serial data events. This type will only appear if you have installed protocol-specific serial data trigger and decode options.

MultiStage Types

MultiStage triggers set one event that "qualifies" or arms, and another that triggers. First select MultiStage to show all the triggers in the group.

QualFirst is only enabled when using Sequence sampling mode. It arms the oscilloscope when the A event occurs in the first segment, then triggers on all subsequent B events, saving each as a Sequence Mode segment.

Qualified arms on the A event, then triggers on the B event. In Normal trigger mode, it automatically resets after the B event, and re-arms upon the next matching A event.

Smart Triggers

Smart triggers allow you to apply Boolean logic conditions to the basic signal characteristics of level, slope, and polarity to determine when to trigger. First select Smart to show all the triggers in the group.

Glitch triggers upon finding a pulse-width that is less than a specified time or within a specified time range.

Window triggers when a signal exits a window defined by voltage thresholds.

Interval triggers upon finding a specific time between two consecutive edges of the same polarity. Use it to capture intervals that fall short of, or exceed, a specified range.

Dropout triggers when a signal loss is detected. The trigger is generated at the end of the timeout period following the last trigger source transition. It is used primarily in Single acquisitions with pre-trigger Delay.

Runt triggers when a pulse crosses a first threshold, but fails to cross a second threshold before re-crossing the first. Other defining conditions for this trigger are the edge (triggers on the slope opposite to that selected) and runt width.

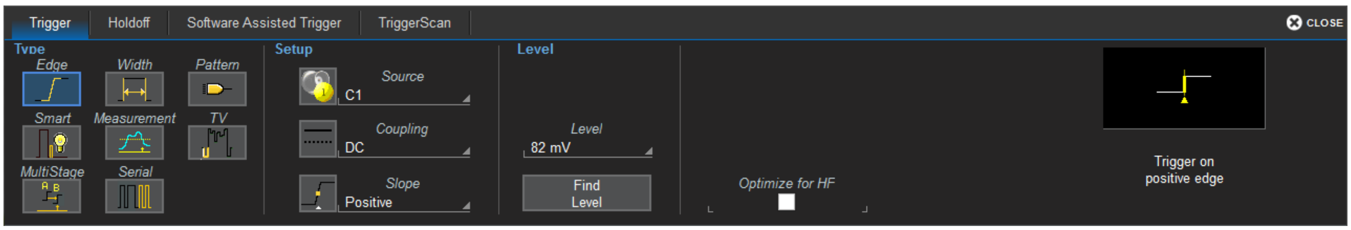
Slew Rate triggers when the rising or falling edge of a pulse crosses an upper and a lower level. The pulse edge must cross the thresholds faster or slower than a selected period of time.

Trigger Set Up

To open the Trigger dialog, press the **front panel Trigger Setup** button or touch the **Trigger descriptor box**.

Different controls will appear depending on the Trigger Type selected (e.g., Slope for Edge triggers). Complete the settings shown after making your selection.

The trigger condition is summarized in a preview window at the far right of the Trigger dialog. Refer to this to confirm your selections are producing the trigger you want.



Source

For most triggers, the **Source** is the analog channel or digital line to inspect for the trigger conditions. Pattern triggers may utilize multiple sources (such as a mix of analog and digital signals), and likewise MultiStage triggers may use different sources for the arming and triggering events.

Tip: On instruments with OneTouch, the trigger source can be easily set by dragging the desired channel descriptor box onto the Trigger descriptor box. Note that the trigger coupling and slope/polarity will revert to whatever was last set on that channel.

Coupling

For analog triggers, specify the type of signal **Coupling** at the input:

- **DC** - Frequency components are coupled to the trigger circuit for high frequency bursts, or where the use of AC coupling would shift the effective trigger level.
- **AC** - Capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
- **LFREJ** - Coupled through a capacitive high-pass filter network, DC is rejected and signal frequencies below 50 kHz are attenuated. For stable triggering on medium to high frequency signals.
- **HFREJ** - DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz (used for triggering on low frequencies).

Slope/Polarity

For some triggers, such as Edge, you will be asked to select the waveform **Slope** (rising vs. falling) on which the triggering event may occur. For others, such as Width, the equivalent selection will be **Polarity** (positive vs. negative).

Level

For analog triggers, enter the voltage **Level** at which the triggering condition must occur. Use the **Find Level** button to set the level to the signal mean.

Trigger types that require multiple crossings to define the triggering condition—such as Window, SlewRate and Runt— will have **Upper Level** and **Lower Level** fields.

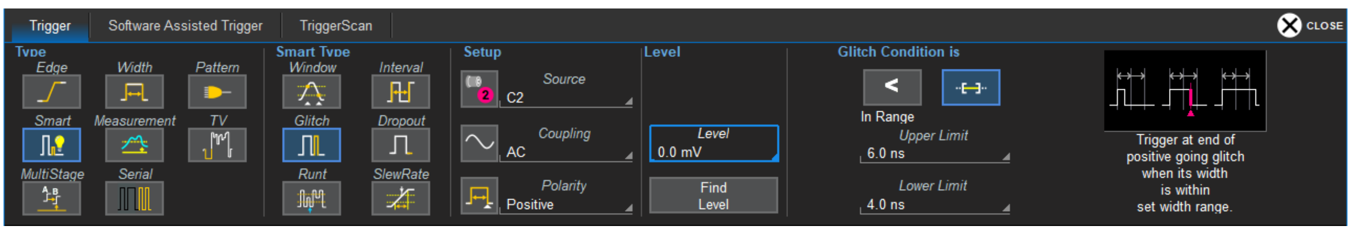
For digital pattern triggers, the level is determined by the **Logic Family** that is set on the digital group. This can also be specified by a custom (User-Defined) crossing **Threshold** and **Hysteresis**. Usually, there will be a separate Logic tab for these settings.

Hysteresis

For Edge triggers, you may select to **Optimize for HF**. When this box is unchecked, the trigger circuit's hysteresis is set quite high to eliminate any potential trigger instability from high-frequency noise riding on a low-frequency input signal. Because HF signal amplitudes tend to be smaller, the Optimize for HF setting reduces the hysteresis value to make the trigger more sensitive. The effect this will have on trigger behavior depends on the signal's spectral and noise content rather than just fundamental frequency, so it's overall effect is difficult to quantify.

Conditions (Smart Triggers)

Smart triggers all allow you to apply Boolean logic to refine the triggering condition beyond simply Level and Slope/Polarity.



The values that satisfy the operators of **Less Than**, **Less Than or Equal To**, **Greater Than**, etc. can be set by entering an **Upper Value** and/or **Lower Value**.

In some cases, it is possible to set a range of values that satisfy the condition. Depending on the trigger, the values may be **In Range** that is bounded by the upper/lower values, or **Out Range**.

The extent of the range can often also be specified by using a **Nominal** and **Delta** value, rather than an absolute upper and lower value. In this case, the Nominal value sets the center of the range, and the Delta determines how many units plus/minus the Nominal value are included in the range.

For Dropout triggers, the default is to **Ignore Opposite Edge**, setting the trigger to dropout of the Positive or Negative edge within the given timeframe. Deselecting it has the effect of setting the trigger to dropout on Both edges.

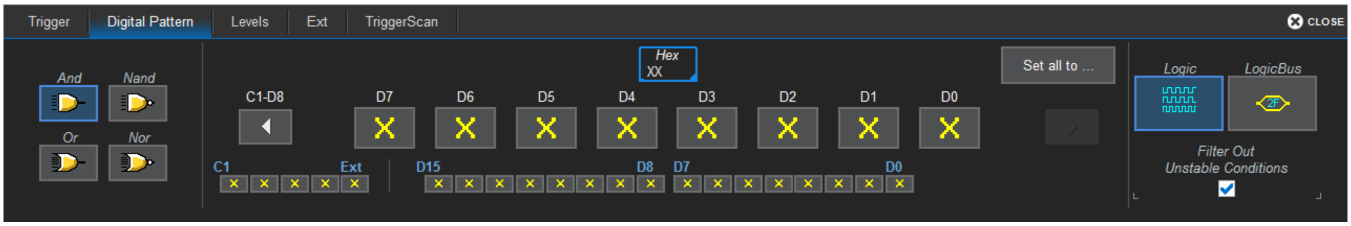
Patterns

A triggering logic pattern may be set on digital lines, analog channels, or a combination of both.

Digital Pattern

A digital pattern is set on a single bus (group) manually or by applying a hexadecimal value, while the remaining lines are disabled ("Don't Care").

The Logic Bus method simplifies pattern set up by utilizing digital groups you have already defined on the Digital Setup dialogs. The size of Hex values will be limited to only those appropriate for the group.



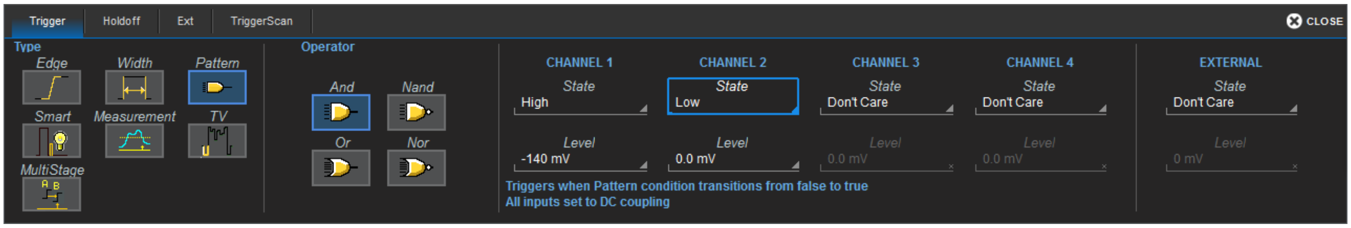
1. On the Trigger dialog, select **Pattern** trigger type. Open the **Digital Pattern** dialog.
2. At the far right of the dialog, choose either **Logic Bus** or **Logic**.
3. Optionally, deselect **Filter Out Unstable Conditions**. This default filter ignores short glitches in logic state triggers that last less than 3.5 ns.
4. If using Logic Bus, touch **Source** and select the digital group. Any lines that are not in this group will now be disabled.
5. To apply a digital logic pattern, either:
    - Enter the hexadecimal value of the pattern in **Hex** or **Value**. Lines will take a logical 1, 0, or X ("Don't Care") according to the pattern. Disabled lines will remain X.
    - For each active line, touch **D*n*** and select whether it must be High or Low compared to the logic threshold. A logical 1 (High) or 0 (Low) now appears on the dialog. Leave X for any line you wish to exclude from the pattern. Use the Left/Right Arrow buttons to display lines in other digital banks.

    Note: As an alternative to 1 or 0, you may set edge conditions on any line. Touch **D*n*** and choose the edge. Edge conditions always assume a logical OR in the overall trigger criteria.

6. If you have not already set a logic threshold, open the **Levels dialog** and select a **Logic Family** for each digital bank from which you've selected lines. To set a custom logic threshold, choose Logic Family User Defined, then enter the **Threshold** voltage and **Hysteresis**.

    Note: Digital lines inherit the Logic Setup made when defining digital groups. However, you can change the logic threshold on the Levels dialog. Logic thresholds can only be set per lead bank, not individual line.

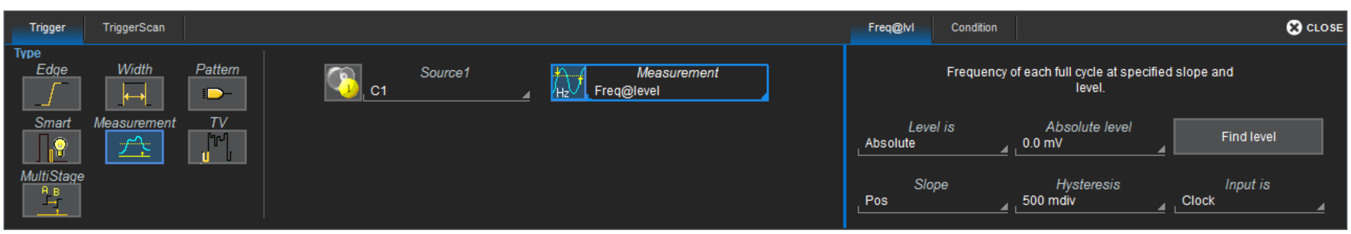
Analog Pattern



1. On the Trigger dialog, select the **Pattern** trigger type.
2. Select the Boolean **Operator** that describes the relationship among analog inputs (e.g., C1 must be High AND C2 must be Low).
3. For each input in the trigger pattern, select what **State** it must be in compared to the threshold Level. Leave "Don't Care" for any input you wish to exclude.
4. For each input included in the trigger, enter the voltage threshold **Level**.
5. If you've included the EXT(ernal trigger) input in the pattern, open the **Ext tab** and enter the trigger **Attenuation**.

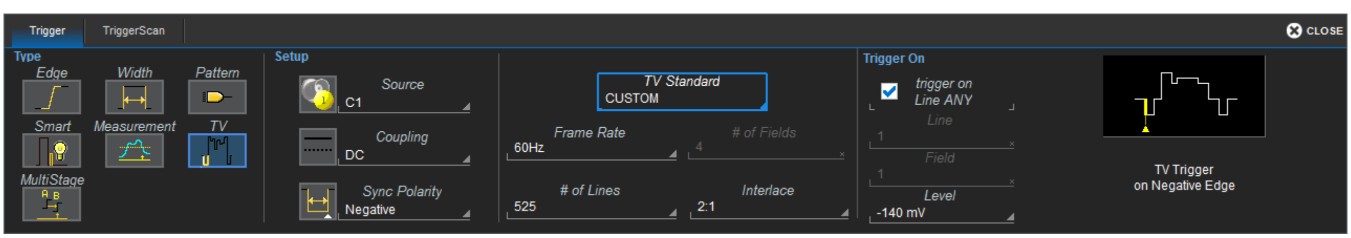
Measurement Trigger

Measurement is not technically a trigger, but permits the instrument to stop and display acquisition based on a post-processing measurement result, similar to a trigger. The available measurements depend on your model instrument and the options installed, and each will offer a different method for specifying the "trigger" conditions, based on the type of measurement. Generally, you will be able to specify a triggering value or range of values, and for @level parameters, the voltage level at which the measurement is taken.



TV Trigger

TV triggers on a specified line and field in standard or custom composite video signals.



1. Choose the **Source** signal input.
2. Choose the signal **TV Standard**. To use a custom signal, also enter the **Frame Rate** , **# of Fields** per

line, **# of Lines**, and **Interlace** ratio.

3. Choose the **Line** and **Field** upon which to trigger.

Serial Trigger

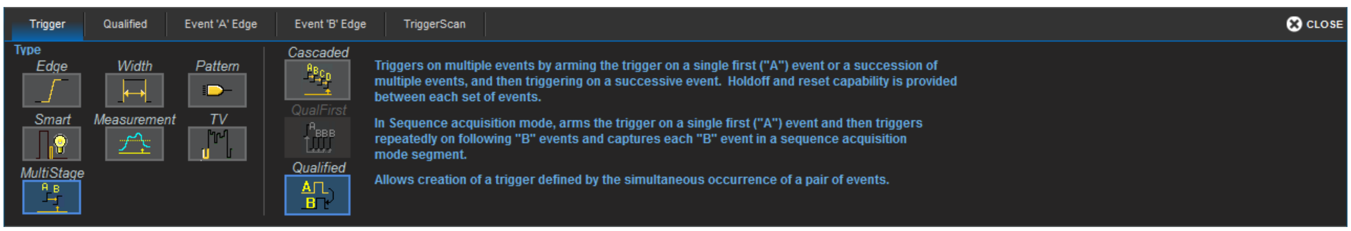
The Serial trigger type will appear if you have installed serial data trigger and decode options. Select the **Serial** type then the desired **Protocol** to open the serial trigger setup dialogs. For setup instructions, see the software instruction manual at teledynelecroy.com/support/techlib under Manuals > Software Options.

Note: Certain software serial triggers can only be input on certain channels; the set up dialog will instruct you which channel to use. Information on the operation of hardware serial pattern triggers can be obtained from the *High Speed Serial Triggers Instruction Manual*.

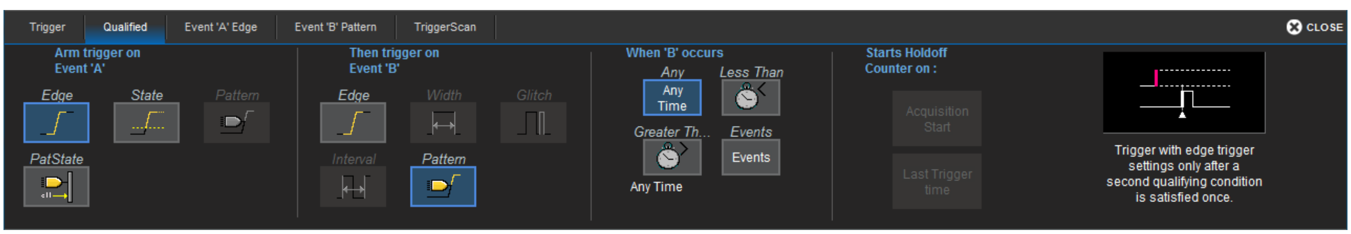
Qualified Trigger

A Qualified trigger arms on the A event, then triggers on the B event. In Normal trigger mode, it automatically resets after the B event, and re-arms upon the next matching A event. Unlike a basic Edge or Pattern trigger with Holdoff, the A and B events can occur in different signals, allowing you to use the state of one signal to "qualify" the trigger on another.

On the Trigger dialog, select **MultiStage** trigger type and choose **Qualified**.



On the **Qualified dialog**, choose the A and B events.



Besides an Edge or Pattern, two special conditions may be selected as the arming event (A):

- **State**, an analog or digital High/Low state ocurring on a single input.
- **PatState**, a pattern of analog or digital High/Low states across multiple inputs.

When B is an Edge or Pattern, use the **When B Occurs** buttons to add a time window to the conditions:

- **Any Time** triggers if B occurs any time after being qualified by A.
- **Less Than** triggers only if B occurs before the time limit once qualified by A.

- **Greater Than** triggers only if B occurs after the time limit once qualified by A.
- **Events** triggers on the next B event after the specified **N Events** once qualified by A.

As with regular Holdoff, the counter may begin from the **Acquisition Start** or the **Last Trigger Time**.

Once you've selected the A and B events on the Qualified dialog, set up the conditions on the respective "Event" dialogs exactly as you would a single-stage trigger.

QualFirst Trigger

The QualFirst trigger, which is only used in Sequence sampling mode, is set up exactly like the Qualified trigger. **QualFirst** arms the oscilloscope when the A event occurs in the first segment, then triggers on all subsequent B events, saving each as a Sequence Mode segment.

Trigger Holdoff

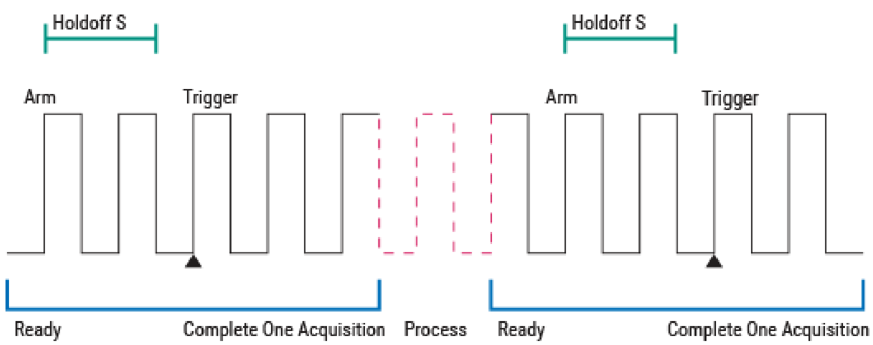
Holdoff is either a period of time or an event count that may be set as an additional condition for Edge and Pattern triggers. Holdoff disables the trigger temporarily, even if the other conditions are met. Use Holdoff to obtain a stable trigger for repetitive, composite waveforms. For example, if the number or duration of sub-signals is known, you can disable them by setting an appropriate Holdoff value.

Note: Qualified triggers operate using time or event conditions similar to Holdoff, but arm and trigger differently.

Hold Off by Time

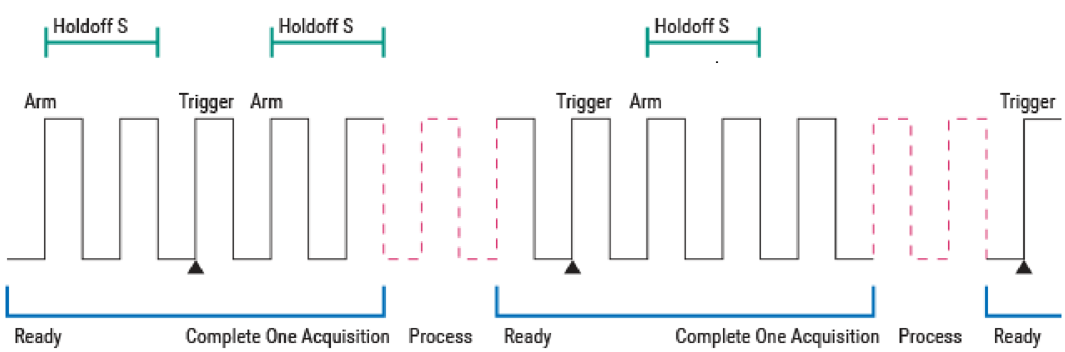
This is a period of time to wait after the arming event before triggering on the next event. The maximum allowed time is 20 seconds; the Holdoff time would otherwise be limited only by the input signal, the coupling, and the instrument's bandwidth.

When a Holdoff by time is counted from the start of the acquisition, the oscilloscope readies, arms on the first event, holds for the specified time, then triggers on the next event. After one full acquisition has completed, the oscilloscope again readies, arms, holds, and triggers for the following acquisition.



Positive Edge trigger with Holdoff by time counted from the start of acquisition.

When a Holdoff by time is counted from the last trigger time, the oscilloscope immediately re-arms on the first event following the trigger and begins counting the Holdoff, rather than wait to complete the full acquisition. The Holdoff count continues even during the very brief time between acquisitions while the oscilloscope is processing. As soon as the Holdoff is satisfied *and* the oscilloscope is again ready, it triggers on the next event. The re-arming and Holdoff may occur in one acquisition, and the trigger in the next.



Positive Edge trigger with Holdoff by time counted from the last trigger time.

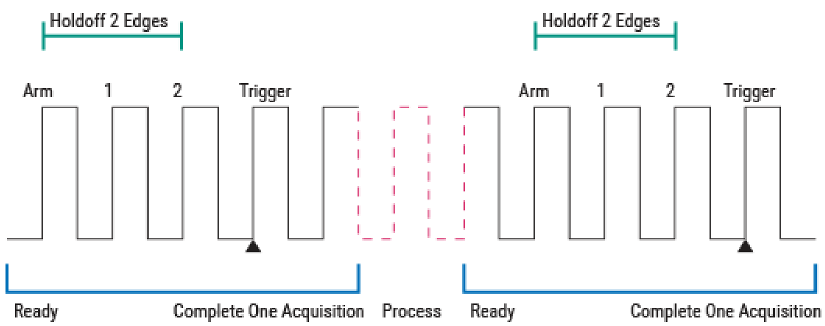
Note: Because there is only one trigger per acquisition, the trigger event will always belongs to the new acquisition. The processing time shown here is for purposes of illustration only.

Regardless of where in the acquisition record the trigger event was found (first edge or last), the display will show time pre- and post-trigger based on your Time/Div and Delay settings.

Hold Off by Events

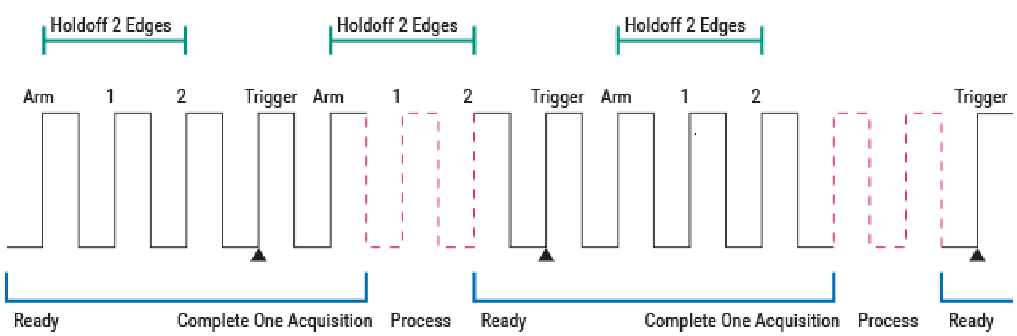
Events refers to the number of times the trigger conditions have been met following the arming event.

For example, if the Holdoff is two edges counted from the start of the acquisition, the oscilloscope readies, arms on the first edge, holds off for the next two, triggers on the fourth edge, then completes the acquisition. Because there must always be a first arming edge, it appears to be "Holdoff plus one."



Positive Edge trigger with Holdoff by events counted from start of acquisition.

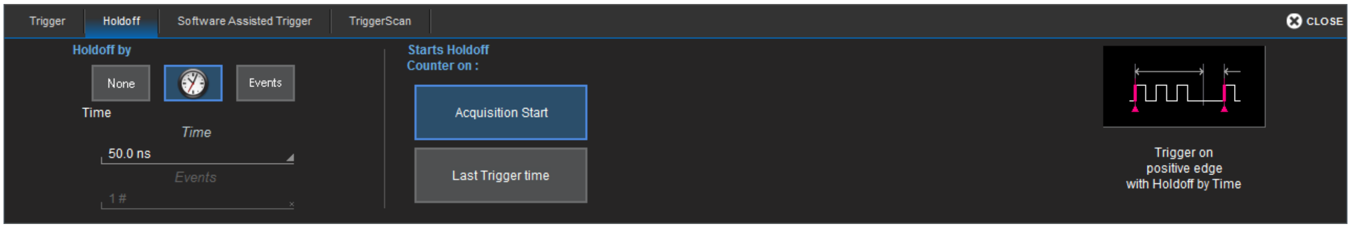
As with Holdoff by time, when a Holdoff by events is counted from the last trigger time, the oscilloscope re-arms immediately following the trigger and begins the Holdoff count. If the count is satisfied by the time the oscilloscope is again ready, the trigger occurs on the next event at the start of the new acquisition.



Positive Edge trigger with Holdoff by events counted from last trigger time.

Holdoff Set Up

To add Holdoff to an Edge or Pattern trigger, touch the Trigger descriptor box or press the front panel Trigger Setup button, then open the **Holdoff tab**.



Choose to **Holdoff by Time** (the clock) or **Events**.

- If using Holdoff by Time, enter the **Time** in S to wait before triggering.
- If using Holdoff by Events, enter the number of **Events** to wait before triggering.

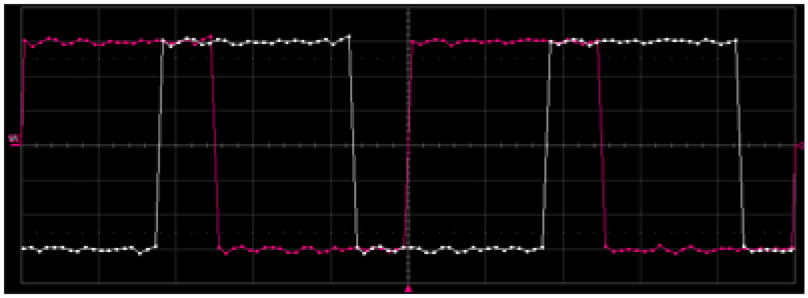
Choose to **Start Holdoff Counter On**:

- Current **Acquisition Start** time.
- **Last Trigger Time**—time of trigger from previous acquisition.

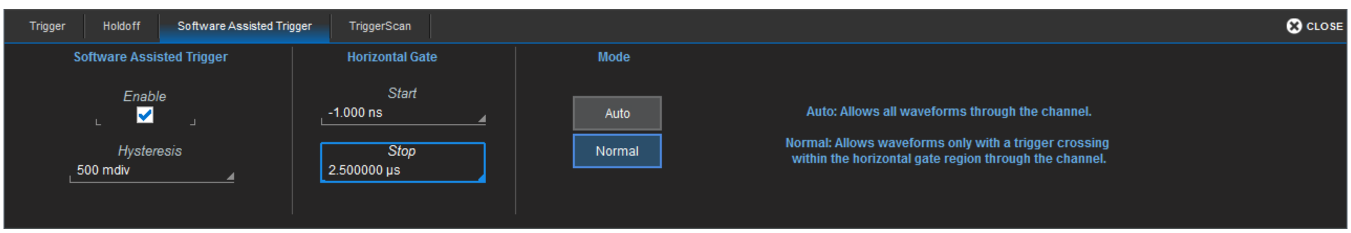
Software Assisted Trigger

Software Assisted Trigger is used to find the trigger-level crossing point closest to the hardware trigger point. It then adjusts the time offset of the waveform so that it is aligned with the specified trigger level and slope.

In the following figure, the white trace is the waveform at the input to software-assisted trigger processing. The red trace shows it after processing, aligned with the trigger point indicated by the red marker below the grid.



Note: This feature can only be used with an Edge trigger type in Normal trigger mode.



]

1. From the menu, choose **Triggers > Trigger Setup**, then touch the **Software Assisted Trigger tab**.
2. Touch **Enable**.
3. Create a trigger window by entering a **Hysteresis** value. This value sets a boundary above and below the main trigger level to exclude noise.
4. Choose **Auto** or **Normal** mode; this determines the trigger behavior when trigger crossings are not found in the trigger source waveform.
   - **Auto** mode allows all waveforms through the channel.
   - **Normal** mode allows waveforms only with a trigger crossing within the horizontal gate region through the channel.
5. Set **Start** and **Stop** time values on the **Horizontal Gate** part of the Software Assisted Trigger tab. These values control where in the waveform the software-assisted trigger processing searches for trigger crossings.

TriggerScan

TriggerScan is a debugging tool that helps you quickly discover waveform anomalies by automating the process of building triggers designed to find rare events in an acquisition.

Triggerscan **trains** the system by looking at acquired waveforms. During the training, TriggerScan analyzes the waveforms to determine what waveforms normally look like. Using this information, it generates a list of smart triggers designed to isolate abnormal situations. All acquisition settings are preserved in the TriggerScan setup file for future use.

After loading the trigger setups from the Trainer, TriggerScan **scans** new acquisitions for rare events by cycling the different triggers. As triggers occur, the waveforms are displayed on the screen. AutoSaving waveform files with each trigger allows you to capture the TriggerScan data.

Training TriggerScan

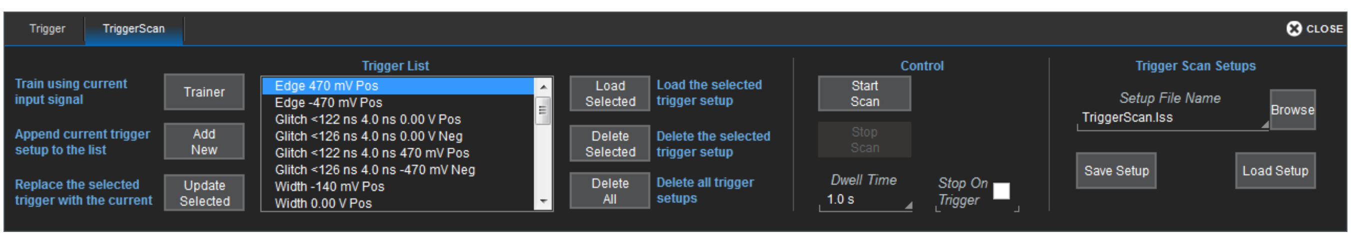
The TriggerScan Trainer inspects the current acquisition and automatically builds a list of trigger setups that could potentially be used to find events of interest.

Note: Run the Trainer if you want to change the trigger types or if you change the channel or signal. You must acquire and display at least 3 cycles of a signal before running the Trainer.

1. Touch **Trigger > Trigger Setup...** from the menu bar, then open the **TriggerScan** tab.
2. Touch the **Trainer** button.
3. On the TriggerScan Trainer pop-up, choose the **Source** channel on which to train and trigger, and select all the trigger types you want to set up.
4. Touch the **Start Training** button. When training is complete, a list of smart trigger setups is displayed in the Trigger List.

Modifying Trigger List

Follow these steps to change the triggers created by TriggerScan. Once you have finalized the Trigger List, you are ready to start scanning.



1. To add a new trigger to the list: Set up the new trigger as desired on the **Trigger** dialog. Then, back on the **TriggerScan** dialog, touch the **Add New** button to append the new trigger to the **Trigger List**.
2. To replace a trigger with one manually set up on the Trigger dialog: highlight the trigger in the Trigger List, then touch the **Update Selected** button.

3. To delete a trigger from the list: highlight the trigger in the Trigger List, then touch the Delete Selected button. All trigger setups can be deleted in one step by touching the Delete All button.
4. Optionally:
   - Enter a Dwell Time the time instrument should wait before loading the next trigger.
   - Check Stop On Trigger. You can use this to isolate events as triggers find them.

Saving TriggerScan Setups

You can preserve the Trigger List by saving it to a setup file. The current Trigger List is not preserved after exiting the application unless you manually save it.

1. On the TriggerScan dialog, touch Setup File Name and enter a file name, or touch the Browse button and select a location and file name.
2. Touch the Save Setup... button.

To reload a saved Trigger List, touch the Browse button, locating the file, then touch Load Setup....

Running TriggerScan

When the Trigger List is set, touch the Start Scan button on the Trigger Scan dialog. The instrument automatically cycles through all the triggers in the list.

To run only a single trigger, select it from the Trigger List and touch Load Selected. The trigger settings are copied to the Trigger dialog and will be used when you start acquisition.

Display

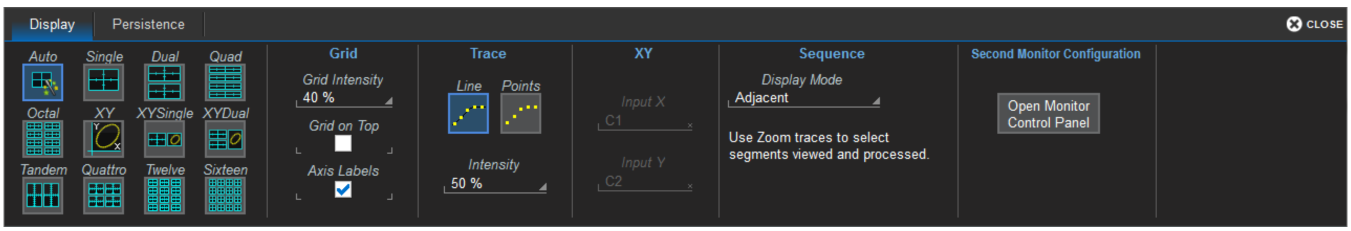
Display settings affect the number and style of grids that appear on screen and some of the visual characteristics of traces, such as persistence.

The oscilloscope features a multi-grid display, where each separate grid represents the full number of 4096 vertical levels. Auto Grid Mode is enabled by default. This feature adds a grid each time a new trace is opened, until no more grids are available. There are display options to show all traces on a Single Grid, or to manually divide the display into different numbers and orientations of grids. The Grid Mode icon shows what the result of the selection will be.

Available with the Q-Scape option, the Q-Scape Display Modes divide the screen into multi-tab displays, each of which can be configured with a different multi-grid display mode.

Display Set Up

To access the Display dialog, choose **Display > Display Setup.**



Q-Scape Display Mode

On instruments with Q-Scape display, the Display Mode determines the number and layout of display tabs. See Q-Scape Display.

Grid Mode

The Grid Mode setting determines the number and layout of display grids, each of which represents the full number of vertical levels. The selection icon shows the number and arrangement of grids.

In Normal Display Mode, select one Grid Mode. In Q-Scape Display Mode, select a Grid Mode for each tab.

Grid Mode	Number	Orientation	Notes
Auto (default)	variable	landscape	Automatically adds or deletes grids as traces turned on/off, up to the maximum supported
Single	1	landscape	All traces share one grid
Dual	2	landscape	One top, one bottom
Tandem	2	portrait	One left, one right
Triple	3	variable	Two landscape grids left, one portrait grid right
Quad	4	landscape	Stacked top to bottom
Quattro	4	landscape	One in each quarter of screen
Octal	8	landscape	Two columns of four stacked top to bottom

Grid Mode	Number	Orientation	Notes
Twelve	12	landscape	Three columns of four stacked top to bottom
Sixteen	16	landscape	Four columns of four stacked top to bottom
XY	1	portrait	Single XY type grid
XYSingle	2	portrait	One VT grid left, one XY grid right

Note: Additional grid modes may become available with the installation of software options.

Grid Intensity

To dim or brighten the background grid lines, touch **Grid Intensity** and enter a value from 0 to 100.

Grid on top superimposes the grid over the waveform.

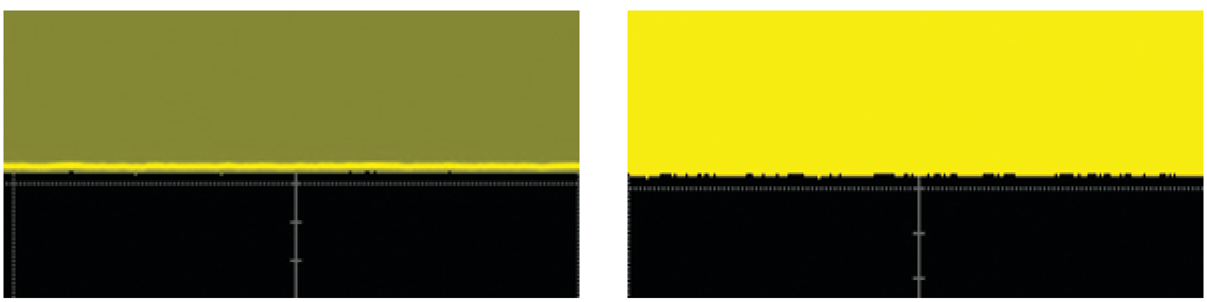
Note: Some waveforms may be hidden from view with the grid on top.

Axis labels display the values represented by each division of the grid, based on your vertical scale and timebase. Turned on by default, they may appear as absolute values or delta from center (0). Deselect the checkbox to remove them from the display.

Trace

Choose a line style for traces: solid **Line** or disconnected sample **Points**.

When more data is available than can actually be displayed given the number of vertical levels, Trace Intensity helps to visualize significant events by applying an algorithm that dims less frequently occurring samples. Touch **Intensity** and enter a value from 0 to 100.



Intensity 40% (left) dims samples that occur ≤ 40% of the time to highlight the more frequent samples, vs. intensity 100% (right) which shows all samples the same.

XY Plots

XY plots display the phase shift between otherwise identical signals. They can be used to display either voltage or frequency on both axes, each axis now corresponding to a different signal input, rather than a different parameter. The shape of the resulting pattern reveals information about phase difference and frequency ratio.

Note: The inputs can be any combination of channels, math functions or memories, but both sources must have the same X-axis scale.

Choose an XY grid mode and select the sources for **Input X** and **Input Y**.

Q-Scape Display

Q-Scape divides the screen into multi-tab displays that can each be configured with different multi-grid display modes. Many of the same functions you can perform in Normal mode, such as moving traces by drag-and-drop, you can also perform in Q-Scape. Q-Scape oscilloscopes have an option for both the Display Mode and the Grid Mode.

Display Mode

The Display Mode setting determines the number and layout of display tabs, each of which can in turn have different numbers and styles of grids within them. Choose either **Normal** (default), a single display with no tabs, or one of the three **Q-Scape display** modes:

Q-Scape Single, a single set of four tabs stacked so that only one at a time is visible. Each tab can have any style of grid mode, same or different.

Q-Scape Dual, two sets of four stacked tabs. Equivalent tabs in each set share the same grid mode, but can display different traces. You can simultaneously view one tab from each set, but they must be different numbered tabs (e.g., Tab 1 from set one and Tab 3 from set two).

Q-Scape Mosaic, a single set of four tabs arranged so that all are viewed simultaneously. Note the tab at the top of each display, which distinguishes Mosaic display mode from Quad grid mode.

In Q-Scape modes, channel, math, memory, and zoom descriptors appear in the tab where the trace is open, while Timebase and Trigger descriptors remain outside the tabs, since they are not associated with any one trace.

Note: If you switch from Normal mode into any Q-Scape mode, all traces will move to Tab 1, from which you can drag-and-drop to redistribute them to other grids. In Q-Scape Single mode, you will not be able to drag-and-drop traces across tabs, only to other Tab1 grids.

Renaming Tabs

The labels shown on the Q-Scape tabs can be changed to something descriptive of what is shown within that display to better orient you to complex screen layouts.

To rename Q-Scape tabs after they open:

1. Touch-and-hold (right-click) on the **Tab *n*** until Rename pops up.
2. Touch **Rename**.
3. Enter the **New Alias**, then touch **OK**.

Extended Display

Note: The "Extend Grids..." option only appears when a second monitor is connected. Minimize the oscilloscope window and use the standard Windows Display controls to make the instrument the primary display. It should be extended, not duplicated.

To take advantage of a second monitor display when in Q-Scape mode:

1. On the Display dialog, select **Extend Grids Onto 2nd Monitor**. The Select Grid pop-up opens.
2. Switch to the **Extended Display** submenu and choose a **Grid** style. Tab 1 and the external monitor will share the same grid style.

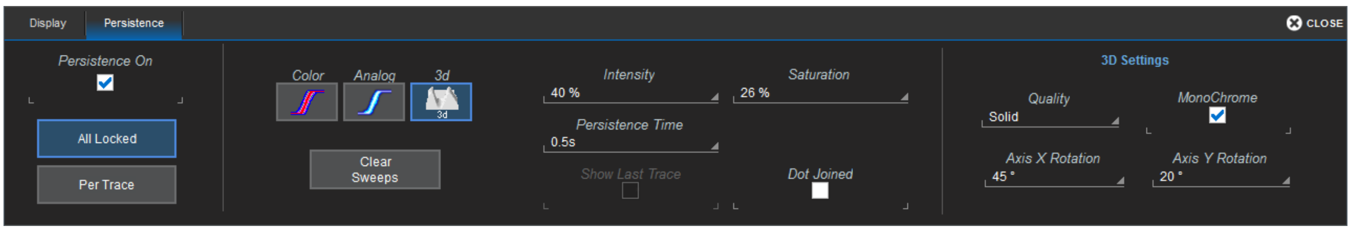
Tip: Drag-and-drop descriptor boxes to place traces on the desired grids.To take full advantage of Q-Scape on the larger display, choose **File > Windowed** on the instrument, then grab the window title bar and drag it to the external monitor screen. Drag the Extended Display window to the instrument screen.

Persistence Display

The Persistence feature retains waveform traces on the display for a set amount of time before allowing them to gradually "decay," similar to the analog-style display of old, phosphor screen oscilloscopes. The display is generated by repeated sampling of events over time and the accumulation of the sampled data into "persistence maps". Statistical integrity is preserved because the duration (decay) is proportional to the persistence population for each amplitude or time combination in the data.

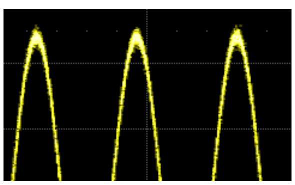
The different persistence modes show the most frequent signal path in three-dimensional intensities of the same color (Analog), or in a graded spectrum of colors (Color).

Access the Persistence dialog from the Display dialog or by choosing **Display > Persistence Setup**.

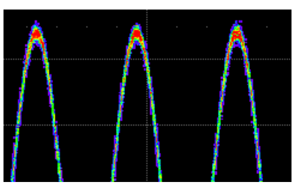


Apply Persistence

1. Check **Persistence On**.
2. To set up all traces together, touch **All Locked**. This constrains all input channels to the same persistence settings. To set up traces individually, touch **Per Trace**.
3. Use the buttons to select a persistence mode:



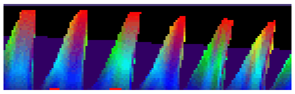
In **Analog Mode**, as a persistence data map develops, different intensities of the same color are assigned to the range between a minimum and a maximum population. The maximum population automatically gets the highest intensity, the minimum population gets the lowest intensity, and intermediate populations get intensities in between these extremes.



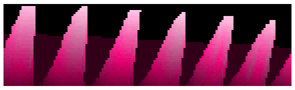
Color Mode persistence works on the same principle as Analog persistence, but instead uses the entire color spectrum rather than intensities of a single hue: violet for minimum population, red for maximum population. In this mode, all traces use all colors, which is helpful for comparing amplitudes by seeking like colors among the traces.

3d Mode persistence creates a topographical view of your waveform from a selection of shadings, textures, and hues. The advantage of the topographical view is that areas of highest and lowest intensity are shown as peaks and valleys, in addition to color or brightness. The shape of the peaks (pointed or flat) can reveal further information about the frequency of occurrences in your waveform. You choose the **Quality** of the 3D display:

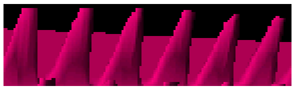
EAR99 technology subject to restrictions on copyrights page.



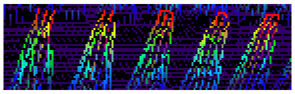
Solid quality can be either color or monochrome. Saturation is set at 50%, with hotter colors indicating highest intensity.



In the monochrome view of solid, the lightest areas indicate highest intensity, corresponding to the red areas in the color view.



In **shaded** (projected light) quality, the shape of the pulses is emphasized. This quality is monochrome only.



In **wire frame** quality, lines of equal intensity are used to construct the persistence map. This display can be either color or **Monochrome**.

In 3d mode, **X Axis Rotation** and **Y Axis Roation** can range 180° from -90° to +90°. In the examples above, the X-axis is rotated 60° an the Y-axis 15°.

> **Tip:** To quickly rotate the graph, grab a corner of the persistence map and drag it in the desired direction.

4. Select the **Saturation** level as a percentage of the total population. All populations above the saturation level are assigned the highest color intensity. At the same time, all populations below the saturation level are assigned the remaining intensities. Data populations are dynamically updated as data from new acquisitions is accumulated. A saturation level of 100% spreads the intensity variation across the entire distribution; at lower saturation levels, the intensity will saturate (become brighter) at a lower population, making visible those events rarely seen at higher saturation levels.
5. In **Persistence Time**, enter the duration of time (in seconds) after which persistence data is erased from the display.
6. You can superimpose the last waveform over the persistence map by selecting **Show Last Trace** (not available in 3d mode).
7. To display persistence traces as a continuous line (instead of a series of sample points), select **Dot Joined**.

Remove Persistence

To turn off persistence and return to the regular trace style, clear the **Persistence On** checkbox.

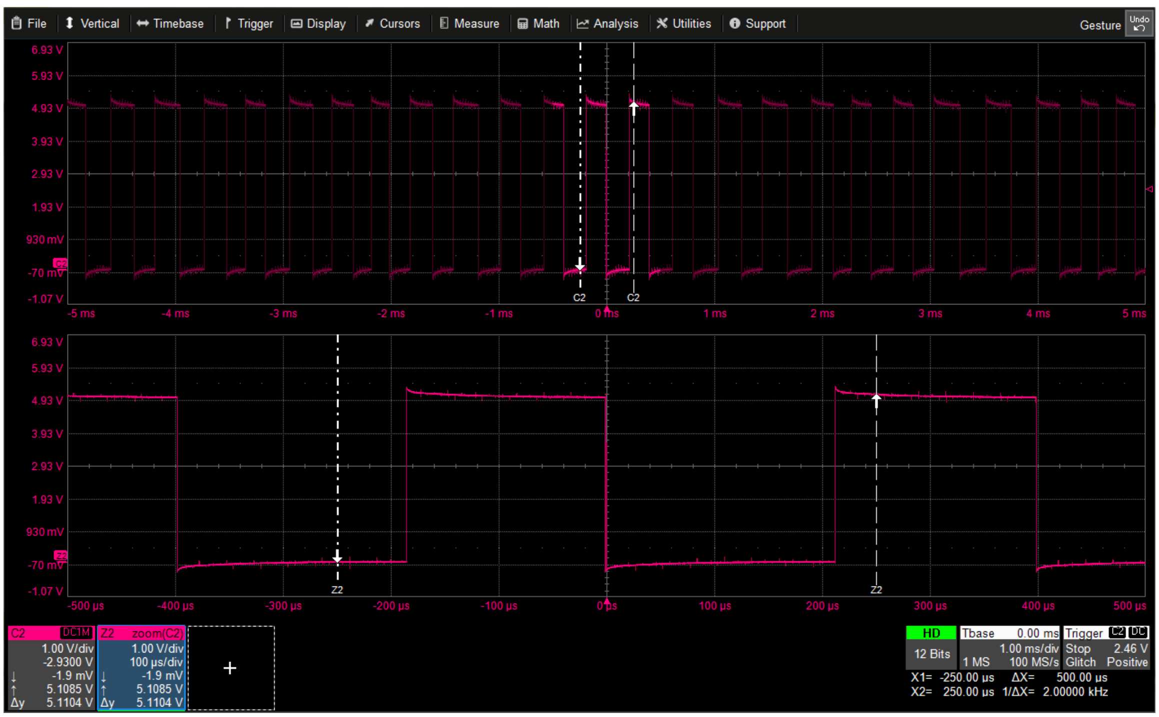
Cursors

Cursors are markers (lines, cross-hairs, and arrows) that identify horizontal and vertical values where they intersect the X or Y axis. Use cursors to make fast, accurate measurements of points on the waveform.

Cursor Types

Horizontal Cursors

Horizontal cursors are positioned at points on the x-axis and will measure the source trace horizontal and vertical values at that point. Usually, these are in units of time and Volts, but may be whatever units are currently configured for the trace. On instruments with OneTouch, they will automatically adjust position to reflect differences in the scale of zooms and source traces when you drag the cursor readout from below the Timebase descriptor box onto the zoom trace grid or descriptor box.



Horizontal Relative Cursors.

The **Horizontal Abs**(olute) cursor displays a single line, X1. The readout below the Timebase and Trigger descriptors shows the horizontal value where X1 intersects the x-axis, in whatever is the x-axis unit of the source trace. The readout on the source trace descriptor box shows the vertical value where X1 intersects the source trace, y@X1, in whatever is the y-axis unit of the source trace.

The **Horizontal Rel**(ative) cursor displays two lines: X1 with the down-pointing arrow, and X2 with the up-pointing arrow. The readout below the Timebase and Trigger descriptors always shows:

- The value where each cursor intersects the x-axis (X1 and X2)
- The difference of X2 – X1 ((Δx)
- For time traces, the frequency in Hz calculated from the delta time (1/(Δx).

When horizontal cursors are not tracking, they can be moved to any position along the x-axis individually. The horizontal delta represents X2 – X1, which will be a positive number so long as X2 remains to the right of X1. If X2 is moved to the left of X1, this will now be a negative number.

The readout on the source trace descriptor box depends on which Show option is selected:

- **Absolute** shows the vertical values where X1 and X2 intersect the source trace.
- **Delta** shows the difference in vertical value where each cursor intersects the source trace (marked by the arrows), calculated as:

  $$y@X2 - y@X1 = \Delta y$$

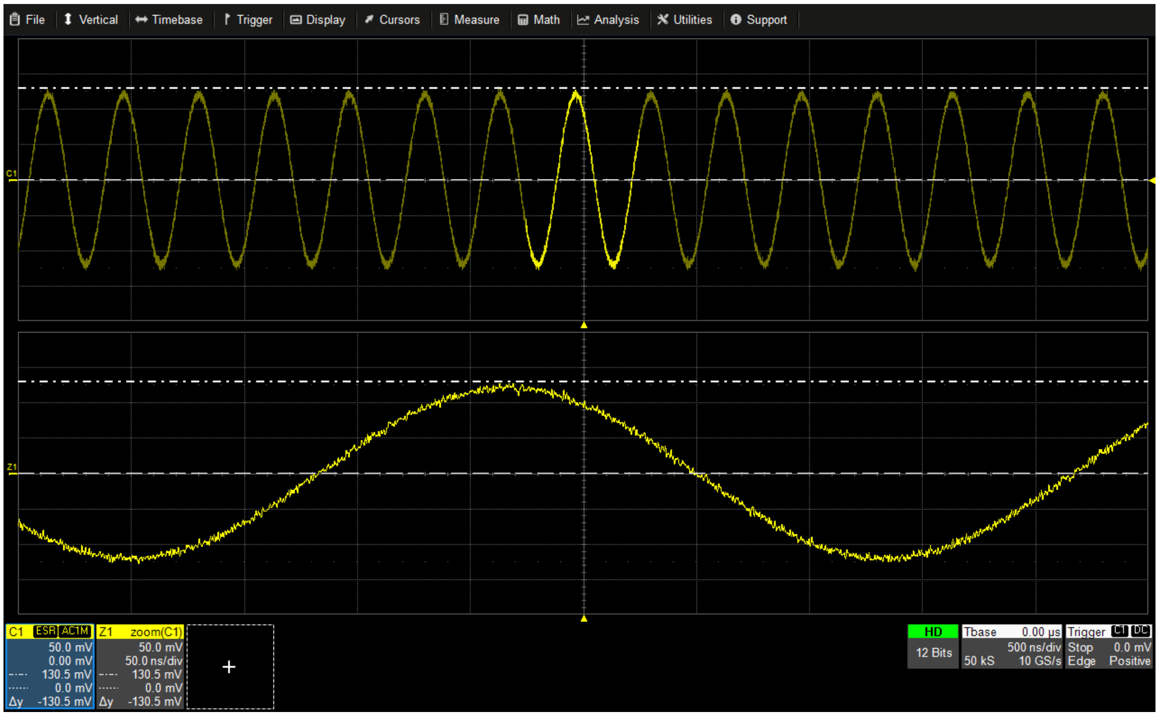
  When the X1 arrow is higher than the X2 arrow, this will be a negative number, as it represents a drop (e.g., in voltage), even when X2 is positioned above the zero level. When the X1 arrow is lower than the X2 arrow, this will be a positive number, as it represents a rise.
- **Abs + Delta** shows both absolute and delta readouts.
- **Slope** calculates the difference in vertical value over the difference in horizontal value represented by the slope from the X1 arrow to the X2 arrow, or: $\Delta y/\Delta x$

  The value will be intelligently converted to whatever unit is appropriate for the source trace. Like the Delta readout, it will be positive or negative depending on the vertical position of X2 relative to X1, regardless of its horizontal position.

Horizontal cursors can be placed on traces whose x-axis has a dimension other than time, such as an FFT or a Trend. When there is at least one non-time-domain trace open, the Standard Cursors dialog contains an **X-Axis** control where you can choose the units to be measured by the horizontal cursors. The options will be appropriate to the types of traces open; for example, if there is an FFT trace, there is an option for Hz. The cursors are placed on those traces that display x-axis values in the selected units, rather than those with an x-axis of time.

Vertical Cursors

Vertical cursors intersect the y-axis and show the vertical value at that point (e.g., a voltage). These cursors can go "off trace" to show vertical scale values that are not represented in the acquisition. Vertical cursors have no horizontal readout below the Timebase descriptor, as they do not have an x-axis element. As they are set by divisions, they remain in the same position and do not "readjust" with changes in the scale of the underlying traces.



Vertical Relative Cursors.

The **Vertical Abs**(olute) cursor displays a single line, Y1. The readout on the source trace descriptor box shows the vertical value at the point where Y1 intersects the y-axis.

The **Vertical Rel**(ative) cursor displays two lines: the dashed-dotted line is Y1, and the dashed line is Y2. As with the Horizontal Relative cursor, the readout on the source trace descriptor box depends on the Show selection:

- **Absolute** shows the vertical values where Y1 and Y2 intersect the y-axis.
- **Delta** shows the difference in value of Y2 – Y1 (Δy). As long as Y2 remains below Y1, this is a negative number, even if Y2 is positioned above the zero level. If Y2 is moved above Y1, it will become a positive number.
- **Abs + Delta** shows both vertical readouts.

Combination Cursors

The **Both Rel**(ative) option places both Vertical Relative and Horizontal Relative cursors together. The vertical readout on the source trace descriptor will be the same as for the Vertical Relative cursor and depends on your Show selection, while the horizontal readout below the Timebase descriptor will be the same as for the Horizontal Relative cursor.

Apply and Position Cursors

To turn on cursors, either:

- From the menu bar, choose **Cursors** and select the desired cursor type from the drop-down list.
- On the front panel, press the **Cursor button** to turn on cursors, then continue pressing to cycle through all the cursor types. Stop when the desired type is displayed.

Note: There must be a trace on the grid for cursors to execute, although acquisition may be in process or stopped when you turn them on.

To turn off cursors, either:

- From the menu bar, choose **Cursors > Off**.
- Continue cycling the **Cursor button** until you reach "Off" (the cursor lines disappear).

To reposition a cursor:

- Drag-and-drop the **cursor marker** to a new position. Indicators outside the grid show to which trace the cursor belongs when you have multiple traces on one grid.

  Use the **Position** data entry controls on the Standard Cursors dialog to place cursors precisely.
- Alternatively, use the Front Panel **Cursor knob**. Push the knob until the correct cursor is selected, then turn the knob to move it. The third press of the Cursor knob selects both cursors so they will track together when the knob is turned.

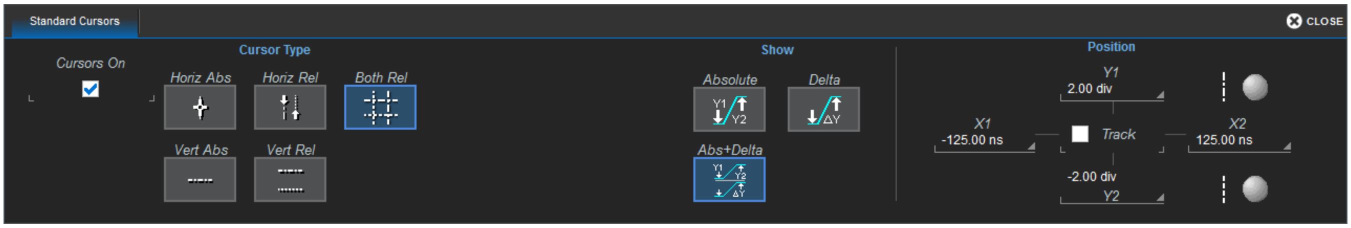
  When there are multiple traces on the same grid, first bring the desired trace to the foreground by touching the trace or its descriptor box. The Cursor knob will only operate on the foreground trace.

On oscilloscopes with OneTouch, if Horizontal cursors are applied to a source trace but do not appear on its dependent traces (e.g., a zoom) because of differences in scale, drag-and-drop the cursor readout from below the Timebase descriptor box onto the target trace descriptor box. This will apply the cursor at the 5 (absolute) or 2.5 and 7.5 (relative) division marks of the target trace and adjust the source trace cursor accordingly.

To track cursors, moving both lines together at a consistent distance, check **Track** on the Standard Cursors dialog. Drag the X1 or Y1 cursor marker, or select the set using the font panel controls and turn the Cursor knob. The delta readouts should show little or no change when tracking, although absolute readouts will change depending on the new position of the cursors. Moving the X2 or Y2 cursor will reset the relative distance and the delta, after which you can again track by moving the X1 or Y1 markers.

Standard Cursors Dialog

These controls can be used instead of the front panel controls to turn on cursors or to refine the cursor position. Access the dialog by choosing **Cursors > Cursors Setup** from the menu bar.



Cursor Type buttons select the type of cursor displayed on the grid.

The **Show** controls determine what values are shown in the vertical readout when using relative cursors:

- **Absolute** shows specific voltages for the two cursor locations.
- **Delta** shows the difference between the specific voltages at the cursor locations.
- **Abs+Delta** shows both the absolute and delta readouts.
- **Slope** (Horizontal Relative only) shows the slope of the waveform between the cursor locations.

Refer to Cursor Types for a detailed explanation of what is shown with each option.

The **Position** controls at the right-side of the Standard Cursors dialog display the current cursor location and can be used to set a new location.

- **X 1** and **X 2** sets the position of Horizontal cursors. They may be entered as time or a fraction of a division.
- **Y 1** and **Y 2** sets the position of Vertical cursors, entered as a fraction of a division.

Track locks relative cursor lines so they move together, maintaining the same distance from each other. Only move X1 or Y1 to reposition the cursors. Moving X2 or Y2 will change the relative distance.

XY Cursors Dialog

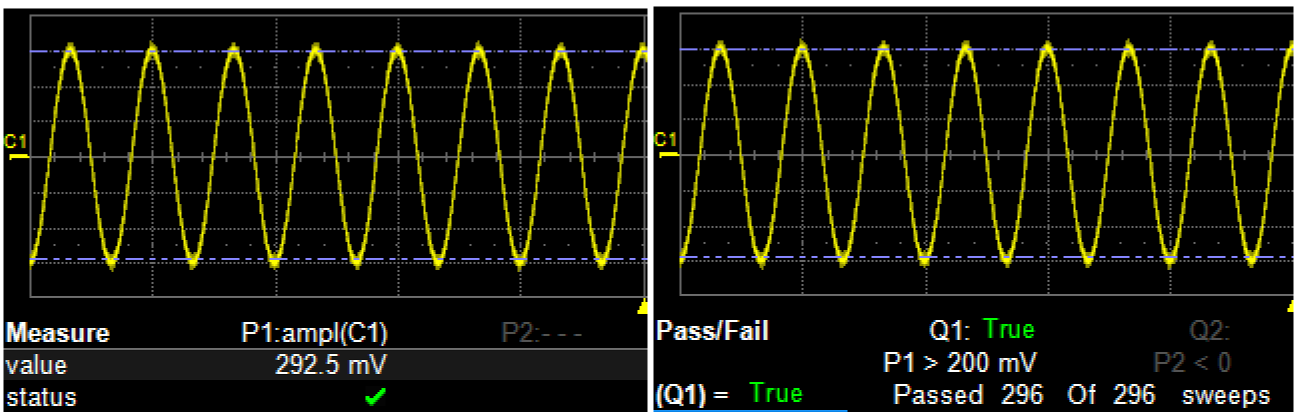
If your Grid Mode selection includes an XY trace display, an XY Cursors dialog appears behind the Standard Cursors dialog. Use it to exactly position and track cursors on XY traces, the same as you use the settings on the Standard Cursors dialog for time traces. The functionality is the same.

Measure

Parameters are tools that give you access to a wide range of waveform properties, such as Rise Time, RMS voltage and Peak-Peak voltage. The instrument offers a quick selection of standard parameters, or you can create a user-defined set of parameters (My Measure) drawn from all available measurements.

On instruments equipped with CustomDSO (XDEV), custom scripts may be used to calculate a parameter. Several scripting "languages" are supported, depending on the application. You can find these options in the Custom submenu of the Measure Selector. Scripts may be imported or written in the instrument's Script Editor window. See Using P Script for a description of the process.

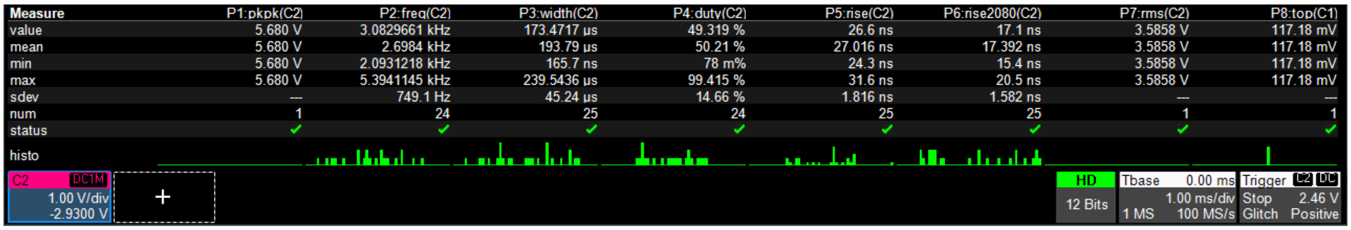
Parameter readouts are shown in a dynamic Measure table that appears below the waveform grids. All active measurements can be used as inputs to other processes, such as Pass/Fail tests, even when the Measure table is hidden from view. The history of a parameter can also be graphed as a histogram, track or trend for statistical analysis.



Measurements are calculated and can be applied to other processes even when they are not displayed. In the image at right, the P1 amplitude measurement is used in a Pass/Fail test, although the Measure table is hidden.

Measure Table

The **value** row of the Measure table shows the measurements taken for each parameter on the last acquisition. You may optionally calculate and display the statistical mean, min, max and sdev of all parameters. Statistics are calculated once per acquisition and accumulate over multiple acquisitions, up to the two billion value limit of the measurement buffer.



Measure	P1:pkpk(C2)	P2:freq(C2)	P3:width(C2)	P4:duty(C2)	P5:rise(C2)	P6:rise2080(C2)	P7:rms(C2)	P8:top(C1)
value	5.680 V	3.0829661 kHz	173.4717 µs	49.319 %	26.6 ns	17.1 ns	3.5858 V	117.18 mV
mean	5.680 V	2.6984 kHz	193.79 µs	50.21 %	27.016 ns	17.392 ns	3.5858 V	117.18 mV
min	5.680 V	2.0931218 kHz	165.7 ns	78 m%	24.3 ns	15.4 ns	3.5858 V	117.18 mV
max	5.680 V	5.3941145 kHz	239.5436 µs	99.415 %	31.6 ns	20.5 ns	3.5858 V	117.18 mV
sdev	---	749.1 Hz	45.24 µs	14.66 %	1.816 ns	1.582 ns	---	---
num	1	24	25	24	25	25	1	1
status	✓	✓	✓	✓	✓	✓	✓	✓
histo								

To quickly set up the table, drag any configured cell onto an empty cell of the Measure table. Then, drag the descriptor box of the trace to be measured onto the new parameter. Touch the cell to choose a new measurement or make any other modifications on the P*n* dialog.

Close setup dialogs when the Measure table is displayed to maximize the area available for viewing waveforms. The table itself can be used to reopen setup dialogs. To open the Parameter Setup (P*n*) dialog, touch the P*n* cell of the table. Touch the far left Measure column to open the Measure dialog and modify the table. The table is static when using standard measurement sets, as these cannot be changed.

Symbols in the **status** row of the Measure table indicate the following:

OK: valid value returned.

Warning: there is a problem with the signal or the setup that prevents measuring. Touch the parameter cell to see an explanation in the message bar.

No Pulse/Insufficient Data: The software is unable to determine top and base. This may indicate that there is insufficient difference between the maximum and minimum for the software to detect a pulse, or there may be an insufficient number of points in the visible top or base of a pulse, such as when closely examining a step response.

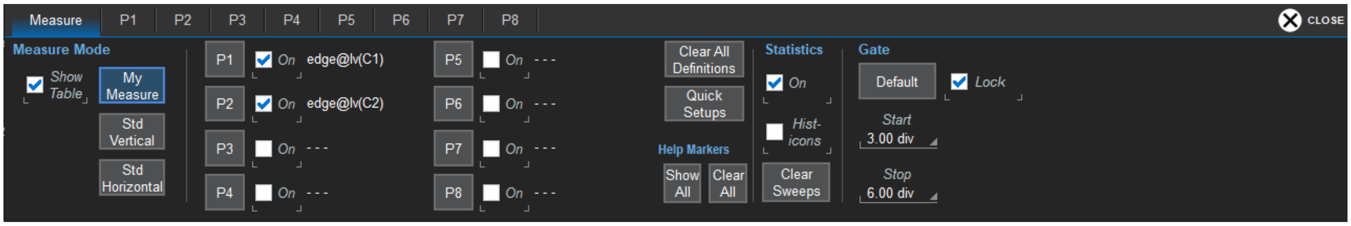
Underflow Condition: The bottom most (negative) sample point of the waveform falls below the ADC range. Probably, the bottom of the pulses appear to be cut off.

Overflow Condition: The top most (positive) sample point of the waveform is above the ADC range. Probably, the top of the pulses appear to be cut off.

Simultaneous Underflow and Overflow Condition: Both conditions are present at once.

Measure Dialog

The Measure Dialog gives quick access to measurement features. Besides turning on/off parameters, use the Measure dialog to display statistics, histicons and help markers for standard and user-defined parameter sets. You can also gate parameters to limit the horizontal scope of the measurement.



Selecting Measurements

1. To open the Measure dialog, touch the **Add New** box and select **Measurement**, or choose **Measure > Measure Setup** from the menu bar.

2. Check **Show Table** to display the readout. This is not required to take the measurement.

3. Choose the **Std. Vertical** or **Std. Horizontal** parameters.

   Or, to turn on your own parameters, choose **My Measure**, then select the **On** checkbox next to each parameter you wish to display. Open the *Pn* dialogs to define new parameters.

4. Optionally:

   - Show Help Markers.
   - Gate parameters to limit measurements to only edges inside the gates.
   - Add Statistics and **Histicons** to the Measure Table.

Touch **Clear Sweeps** to reset all measurement counters and restart all statistics.

Touch **Clear All Definitions** to reset all parameters to "None".

Caution: Definitions cannot be restored after clearing, you must repeat parameter set up.

Standard Parameter Sets

The pre-configured standard parameter sets available on the Measure dialog are:

- **Std Vertical**: Pk-Pk, Ampl, Max, Min, Sdev, Mean, Base, Top
- **Std Horizontal**: Rise, Fall, Period, Frequency, Width, Duty Cycle, Delay, N Points

To quickly begin a new set of user-defined parameters based on the standard vertical or horizontal sets, touch **Quick Setups** and choose the set to copy. Refine the settings on the *Pn* dialogs.

Statistics and Histicons

Checking **Statistics On** on the Measure dialog adds the mean, min, max and sdev of each parameter to the measured value shown on the Measure table.

Statistics for each parameter are calculated once per acquisition and accumulate until you either Clear Sweeps or the measurement buffer is full. The Num row of the Measure table shows the total number of measurements included in the Statistics calculation. If the measurement is gated, the statistics are calculated for only the data points between the gates, just as the parameter value itself will reflect the limits imposed by the gate.

Mean	The weighted mean of the parameter calculated over the number of times shown.
Min	The minimum value of the parameter measured over the number of times shown.
Max	The maximum value of the parameter measured over the number of times shown.
Sdev	The population standard deviation of the parameter calculated over the number of times shown.
Num	For any parameter that computes once on an entire acquisition, Num represents the number of sweeps over which the statistics are computed. For any parameter that computes on every event within an acquisition, such as a full period, Num represents the number of events per sweep times the number of sweeps computed. Thus, for a Single acquisition of five periods, the Num shown for any per period measurements will be 5, as five measurements were made and the statistics reflect those five measurements. After another Single acquisition, Num will be 10, or five measurements times two sweeps. The statistics now reflect all 10 measurements.

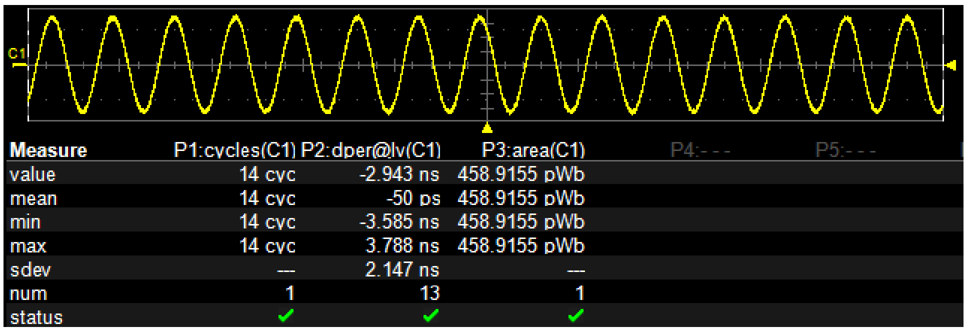
Histicons are miniature histograms of parameters that may be added to the Measure table readout. These thumbnail histograms let you see at a glance the statistical distribution of each parameter. Check **Histicons** on the Measure dialog.

Tip: You can quickly display a full histogram by touching the histicon you want to enlarge.

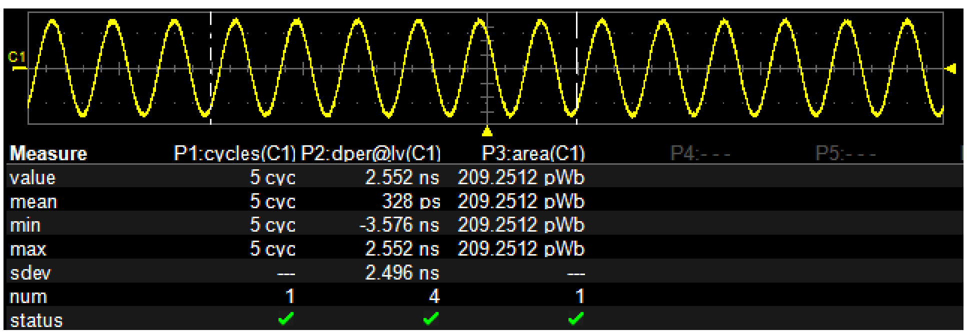
Gating Measurements

All measurements are calculated on only that portion of the waveform trace that is visible on the grid *and* within the measurement gates. Any setting that moves the trace outside the observation window or makes it appear "clipped" will affect measurements.

The default starting positions of the measurement gate posts are 0 div and 10 div, which coincide with the left and right edges of the grid, and the First and Last points. Therefore, the measurement gates initially enclose the entire visible acquisition. By moving the measurement gates, you can focus the measurement on the section of the acquisition of greatest interest. For example, if you "gate" six rising edges of a waveform, calculations are performed only on the six pulses bounded by the gate posts.



Measure	P1:cycles(C1)	P2:dper@lv(C1)	P3:area(C1)	P4:- - -	P5:- - -
value	14 cyc	-2.943 ns	458.9155 pWb		
mean	14 cyc	-50 ps	458.9155 pWb		
min	14 cyc	-3.585 ns	458.9155 pWb		
max	14 cyc	3.788 ns	458.9155 pWb		
sdev	---	2.147 ns	---		
num	1	13	1		
status	✓	✓	✓		



Measure	P1:cycles(C1)	P2:dper@lv(C1)	P3:area(C1)	P4:- - -	P5:- - -
value	5 cyc	2.552 ns	209.2512 pWb		
mean	5 cyc	328 ps	209.2512 pWb		
min	5 cyc	-3.576 ns	209.2512 pWb		
max	5 cyc	2.552 ns	209.2512 pWb		
sdev	---	2.496 ns	---		
num	1	4	1		
status	✓	✓	✓		

Gates (vertical white lines) limit the data measured and visibly change measurement calculations.

If you wish to gate all parameters to the same number of edges, use the **Gate** fields on the Measure dialog. For user-defined parameter sets ("My Measure"), first check **Lock** to activate the Start and Stop fields. Standard parameter sets are automatically gated to the same number of edges, you do not need to first Lock the gates. The quickest way to set a gate is to drag the gate posts from the far left and right of the grid to the desired positions, or you can enter the **Start** and **Stop** positions down to hundredths of a division. When gates are locked, all measurements will change together when you drag either gate post to reposition the gate.

Deselect Lock to return the gates to their last unlocked position. Touch **Default** to return the gates to the width of the grid.

To apply different gates to individual user-defined parameters, deselect Lock on the Measure dialog and use the Gate subdialog of the P*n* dialog to enter the Start and Stop positions. These positions will be resumed when gates are unlocked.

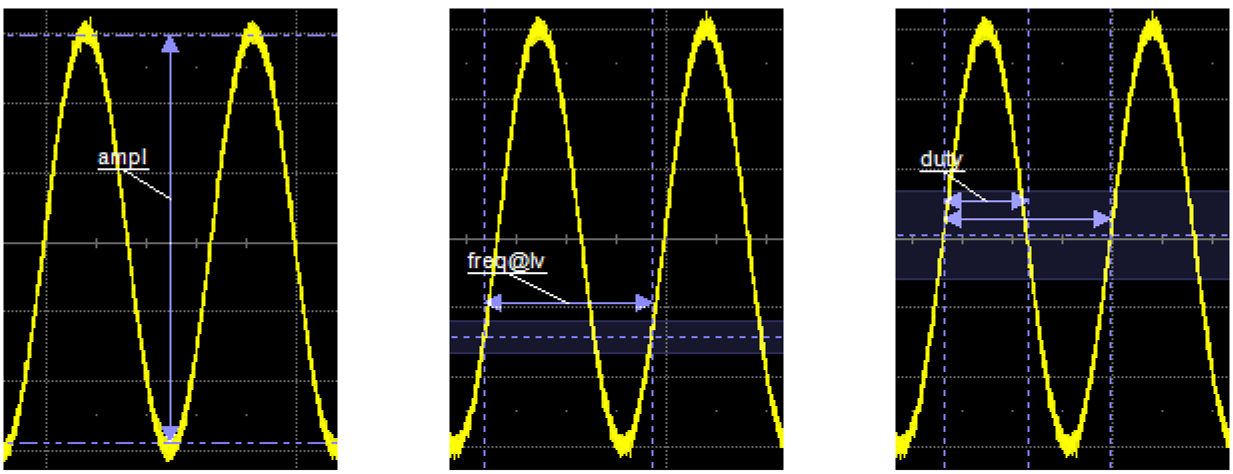
Help Markers

Help Markers clarify measurements by displaying lines, labels and hysteresis bands to mark the points being measured on the trace. For "@Level" parameters, markers make it easier to see where your waveform intersects the chosen level. If you change the set of parameters displayed, the markers will change, as well.

You can choose to use **Simple** markers, which are only the lines, or **Detailed** markers, which include the measurement point labels.

You also have the option, by means of the **Always On** checkbox, to leave the markers displayed over traces after you have closed the Measure dialogs or readout table.

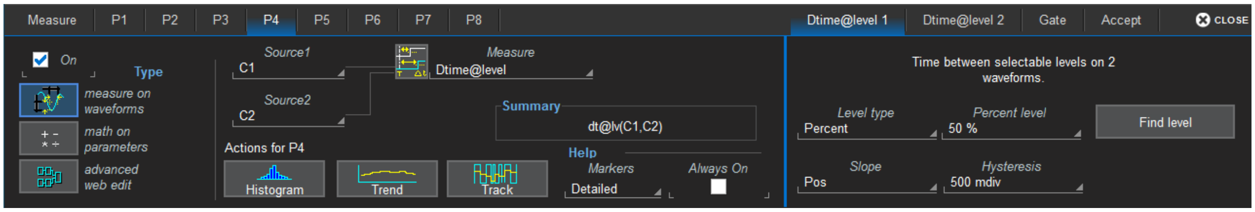
Note: Unlike regular cursors, which are white and can be moved, help markers are blue and only augment the display; they cannot be moved, and they do not reset the measurement points. Some optional analysis software packages include markers designed specially for that domain of reference, which are documented in the option manual.



Detailed Help Markers.

Parameter Set Up

Use the P*n* dialogs to define a new set of measurement parameters ("My Measure").



1. From the menu bar, choose **Measure > Measure Setup**.
2. Choose Measure Mode **My Measure**.
3. Open the **P*n*** dialog and check **On** to activate the parameter.
4. Select the Type **Measure On Waveforms**.
5. Touch the **Source** selectors and choose the trace to be measured. Some measurements require two sources.
6. Touch the **Measure** field and select the measurement from the pop-up menu.
7. Make any further selections on the subdialogs that appear after your Measure selection. These are explained on the dialog and are necessary to fully define the measurement.
    - For @Level parameters, enter the measurement level and optional hysteresis.
    - For certain time-based measurements, choose the slope(s) on which to begin the measurement.
    - Where relevant, choose to make the measurement per period or cycle.
8. Optionally, turn on help markers, or use the parameter subdialogs to:
    - Gate measurements (limit the horizontal scope of the measurement)
    - Accept only measurements that meet additional conditions
    - Change the output units of measure.

Slope, Level and Hysteresis

Several time-based measurements enable you to select on which edge of the waveform the measurement begins, or at what level the measurement is taken (e.g., *@level parameters). These measurements usually also offer options for determining how the level is defined, and for setting up a hysteresis band to prevent unwanted waveform effects from altering the measurement.

Slope

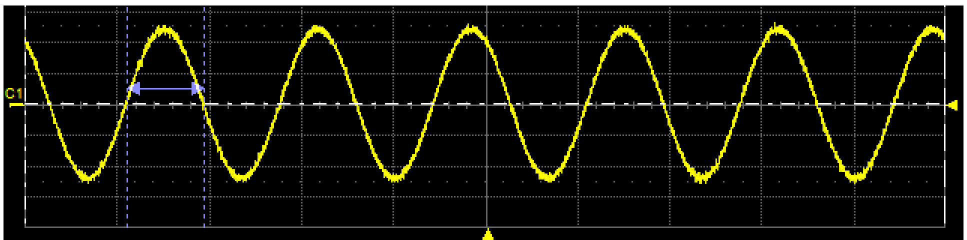
The Slope determines the edge of a pulse (the X axis point) at which measurements are made, or from which edge to which edge a single period/cycle is counted. Slope can impact per period and cyclic measurements, as the number of complete periods may be different depending on which edge is used.

For two-input parameters, such as Dtime@level, you can specify the Slope used for each input separately to determine the points on each waveform to be compared.

There are three possible settings for Slope:

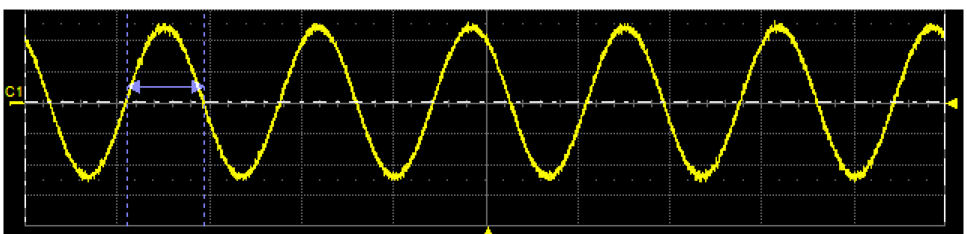
- **Positive**, use the rising edge of a pulse
- **Negative**, use the falling edge of a pulse
- **Both**, use either edge, whichever next occurs that meets the other criteria

Here, the P1 Half Period parameter uses a Slope of Both, resulting in 10 measurements in this five period acquisition, one every time the signal crosses the 50% Level, rising or falling (only the first rising edge measurement is marked by the Help Marker).



Measure	P1:hper(C1)	P2:period(C1)	P3:- - -
value	167.771 ns	333.513 ns	
mean	166.551 ns	333.095 ns	
min	164.668 ns	330.784 ns	
max	168.888 ns	335.185 ns	
sdev	1.410 ns	1.449 ns	
num	10	5	
status	✓	✓	

If Slope were set to Positive, there would be only five measurements in this acquisition, one every time the rising edge crossed the 50% Level, but not the falling edge (the Num row shows the difference).



Measure	P1:hper(C1)	P2:period(C1)	P3:- - -
value	165.781 ns	333.513 ns	
mean	165.744 ns	333.095 ns	
min	164.668 ns	330.784 ns	
max	168.147 ns	335.185 ns	
sdev	1.274 ns	1.449 ns	
num	5	5	
status	✓	✓	

Level

Level is the point on the Vertical (Y) axis used to take a measurement. The default Level for measurements is 0 V or 50% amplitude, the center grid line when there is no Vertical Offset . When Level can be specified, it can usually be set as either:

- An **Absolute** number of Volts, or whatever is the vertical unit of the source waveform
- A **Percent** of waveform amplitude, as determined by the Top-Base calculation

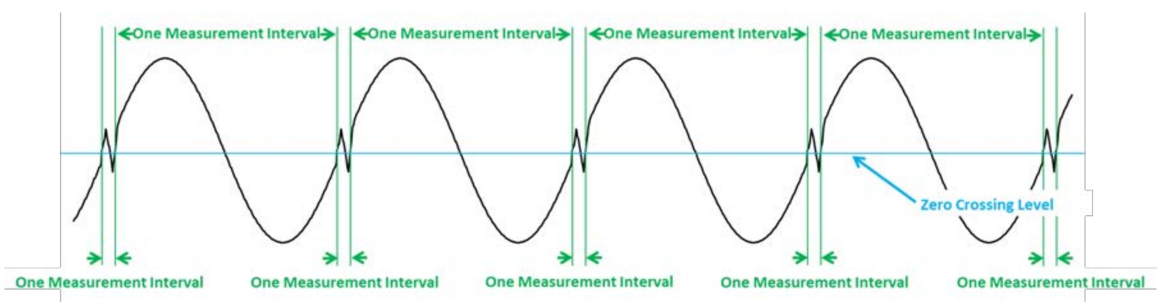
Measurements that may be calculated per period usually have a Levels setting to specify the Level on the rising edge from which full periods are counted.

Hysteresis

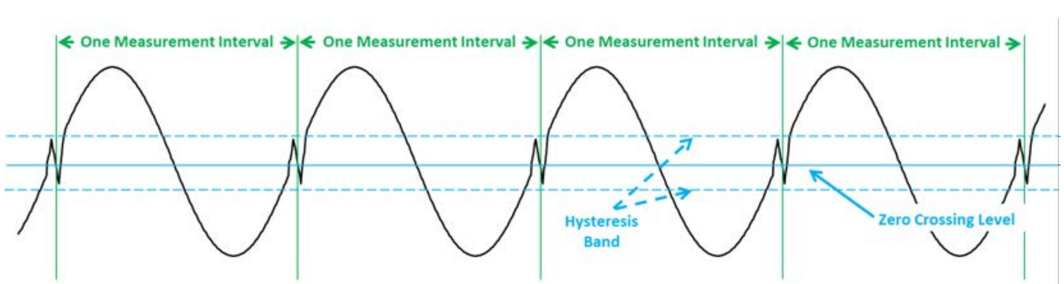
The Hysteresis setting defines an amplitude "band" which the input signal must exceed before a crossing can be determined. Some non-zero Hysteresis value is required to prevent false crossing determinations. The default value is 500 millidivisions (mdiv), with the unit "divisions" being equal to oscilloscope vertical grid divisions.

- Lower hysteresis values improve the ability to detect crossings on a smaller amplitude signal, but with risk that false crossings will be detected. This also means that the signal must meet a minimum amplitude requirement, and be relatively noise free at lower amplitudes.
- Higher hysteresis values improve the ability to reject the impact of signal distortion or noise in determination of the crossing, but with risk that accuracy of crossing detection will be reduced.

For example: a non-monotonic signal crosses the 50% measurement threshold on the rising edge nine times within the observation window, although there are actually only four waveform cycles. Without Hysteresis, the non-monotonicity period is incorrectly detected as crossing and therefore a measurement interval, which will result in incorrect calculation of periodic measurements.



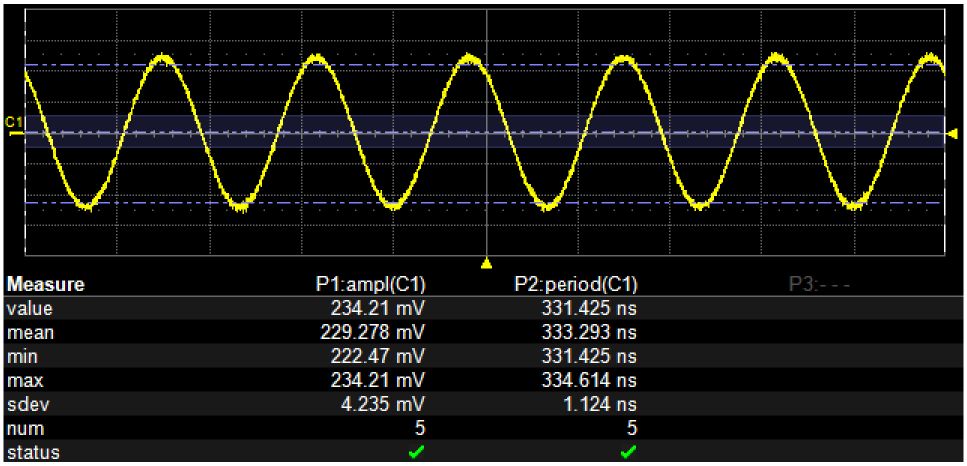
A Hysteresis level that is greater than the amplitude of the non-monotonicity results in only four measurement intervals, avoiding false measurement calculations.



Periodic and Cyclic Measurements

Some measurement set up dialogs display a checkbox for Show one value per period or Cyclic.

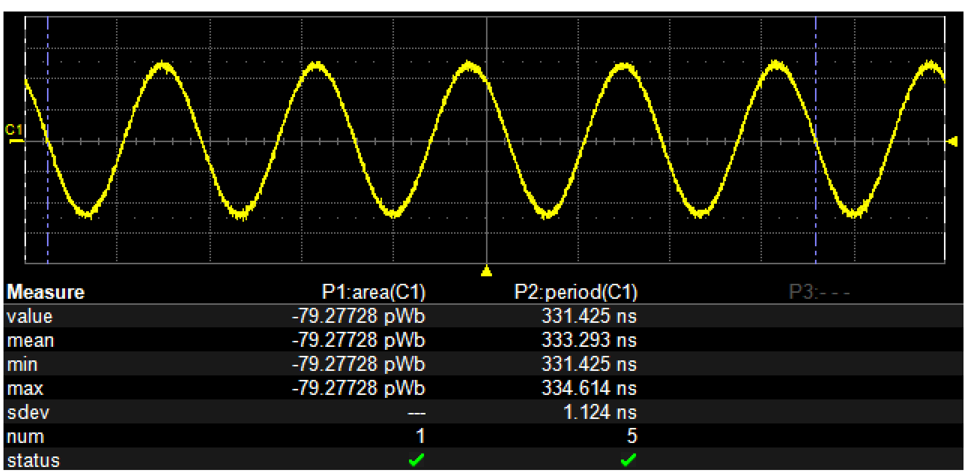
Per period measurements compute one measurement per full waveform period in the acquisition. The value returned is the mean of all the per period measurements. Usually, per period is an option for Vertical parameters, like Amplitude, that would normally compute only one measurement per acquisition. The example below shows how the P1 Amplitude parameter Num statistic changes from 1 to 5 when set to measure per period. A hysteresis band Help Marker also appears, showing the level at which the periods are counted.



Measure	P1:ampl(C1)	P2:period(C1)	P3:- - -
value	234.21 mV	331.425 ns	
mean	229.278 mV	333.293 ns	
min	222.47 mV	331.425 ns	
max	234.21 mV	334.614 ns	
sdev	4.235 mV	1.124 ns	
num	5	5	
status	✓	✓	

Tip: A WaveScan set to Measurement mode on the same source and measurement will show all the per period values within the acquisition.

Similarly, cyclic measurements are computed over the span of full periods in the acquisition, ignoring any partial periods at the beginning or end of the acquisition. For example, if the Area parameter is set to Cyclic, and the acquisition has five full cycles, then only the Area of those five cycles is calculated, as shown below. The blue Help Markers show the five full periods, while the partial periods at the start and end of the acquisition are omitted. Normally, the Area of the full, visible acquisition is calculated. In both cases, only one measurement value is returned, but the value will be different.

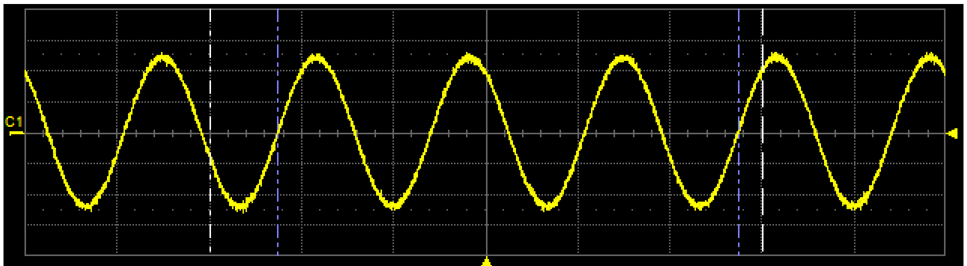


Measure	P1:area(C1)	P2:period(C1)	P3:- - -
value	-79.27728 pWb	331.425 ns	
mean	-79.27728 pWb	333.293 ns	
min	-79.27728 pWb	331.425 ns	
max	-79.27728 pWb	334.614 ns	
sdev	---	1.124 ns	
num	1	5	
status	✓	✓	

Counting Periods/Cycles

Per period and cyclic measurements are subject to the other measurement settings, such as the Slope and Level from which to count the waveform periods. Always visit the subdialogs that appear when checking Show one value per period or Cyclic to determine if there are further settings to be made.

When the measure gates are in their default positions of 0 and 10 div, the periods/cycles are counted across the full width of the grid. If the gates are set elsewhere, then the measurements are computed on only as many full periods/cycles are within the gate. As shown below, gating the cyclic Area measurement reduces the calculation to the three full cycles that are within the measure gates.

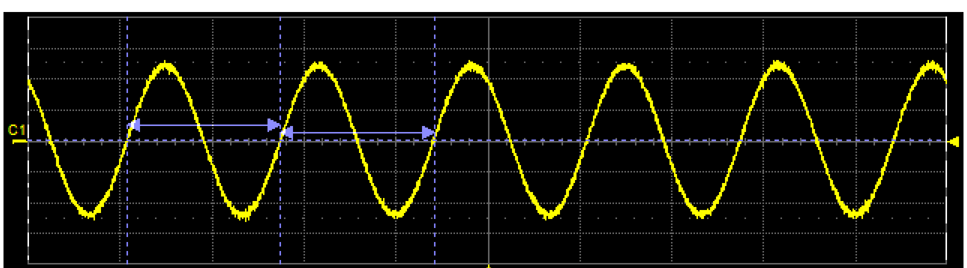


Measure	P1:area(C1)	P2:period(C1)	P3:- - -
value	46.66824 pWb	331.425 ns	
mean	46.66824 pWb	333.293 ns	
min	46.66824 pWb	331.425 ns	
max	46.66824 pWb	334.614 ns	
sdev	---	1.124 ns	
num	1	5	
status	✓	✓	

N-Cycle and N-Period Set Up

The measurements Period@level and Dperiod@level include settings to specify how often to make the measurement and at which cycle to start, rather than using only the full integer number of cycles. The subdialogs containing these settings are called variously N-Cycle Setup and N-Period Setup, but they do the same thing.

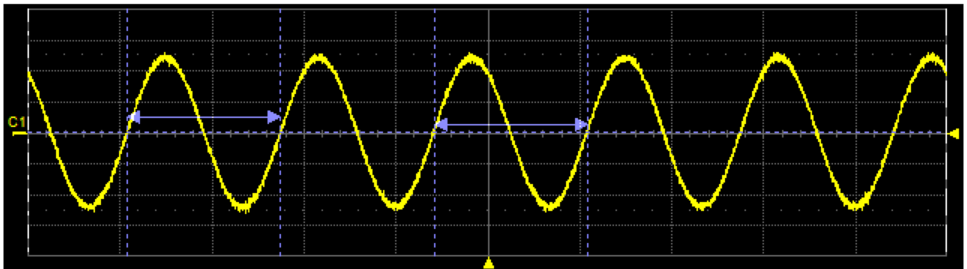
Group Size sets the interval of full cycles at which to take the measurement. For example, the image below shows an acquisitive of five full cycles, which is evident from the P2 Period statistics showing Num 5, indicating that's how many periods were calculated. P1 Dperiod@level, which by default measures the delta between two consecutive periods, shows only a Num of 4, because using a Group Size of 1, four comparisons could be made—period 1 to period 2, period 2 to period 3, and so forth.



Measure	P1:dper@lv(C1)	P2:period(C1)	P3:- - -
value	-1.686 ns	331.425 ns	
mean	-401 ps	333.293 ns	
min	-1.686 ns	331.425 ns	
max	1.253 ns	334.614 ns	
sdev	1.239 ns	1.124 ns	
num	4	5	
status	✓	✓	

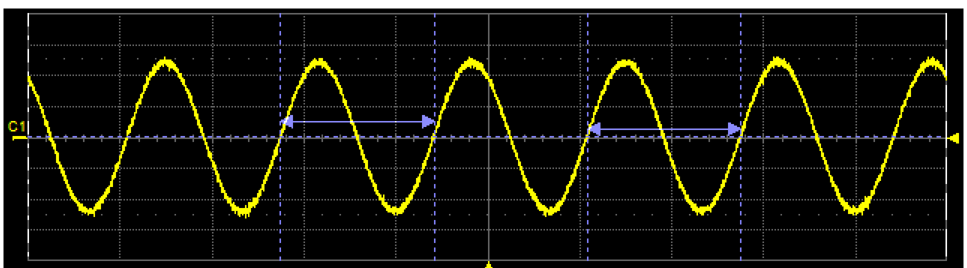
Note that although the Help Markers overlay the first two periods, the measurement value of -1.686 ns reported is actually the final measurement comparing the last two periods, as would be shown by looking at a WaveScan of the same acquisition.

When the Dperiod@level Group Size is changed to 2, the measurement is taken every other period, and because only two measurements can be made out of the five period total (periods 1 to 3 and 2 to 4), the P1 Num changes to 2.



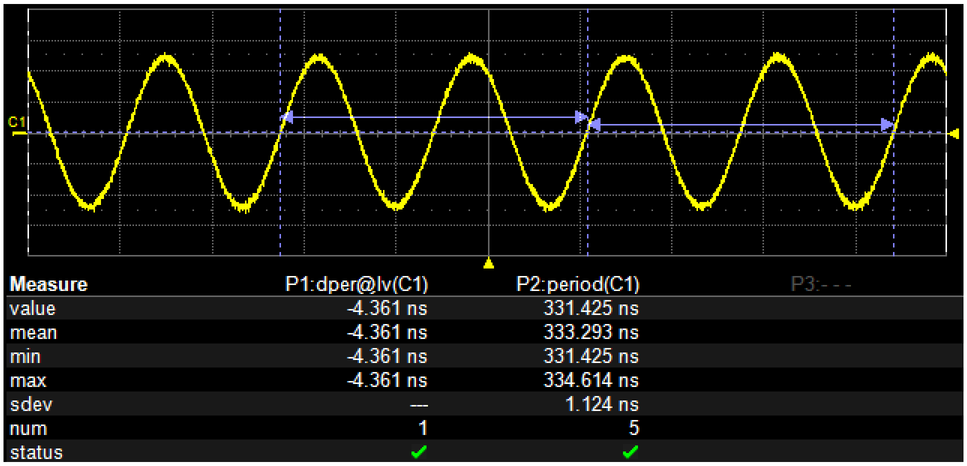
Measure	P1:dper@lv(C1)	P2:period(C1)	P3:- - -
value	-3.189 ns	331.425 ns	
mean	-803 ps	333.293 ns	
min	-3.189 ns	331.425 ns	
max	1.584 ns	334.614 ns	
sdev	2.386 ns	1.124 ns	
num	2	5	
status	✓	✓	

Start Cycle is the cycle from which to begin counting the measurement, which can be any number *Group Size*-1, as the count will begin with the first full group. When the P1 Dperiod@level Start Cycle is changed from 0 to 1, period 1 is skipped, and the measurement begins at period 2. Now, Num has changed to 1, because only one measurement can be made using periods 2 and 4, omitting 3 and 5 due to the Group Size of 2.



Measure	P1:dper@lv(C1)	P2:period(C1)	P3:- - -
value	-1.172 ns	331.425 ns	
mean	-1.172 ns	333.293 ns	
min	-1.172 ns	331.425 ns	
max	-1.172 ns	334.614 ns	
sdev	---	1.124 ns	
num	1	5	
status	✓	✓	

Dperiod@level also allows you to also specify how the Group Size interval is treated, as a separator or a summation. Use One Value Per Group (shown above) measures every *n*th cycle, the first in each group. Sum All Values in Group (shown below) calculates the sum of all the cycles in the group and compares it to that of the next group.

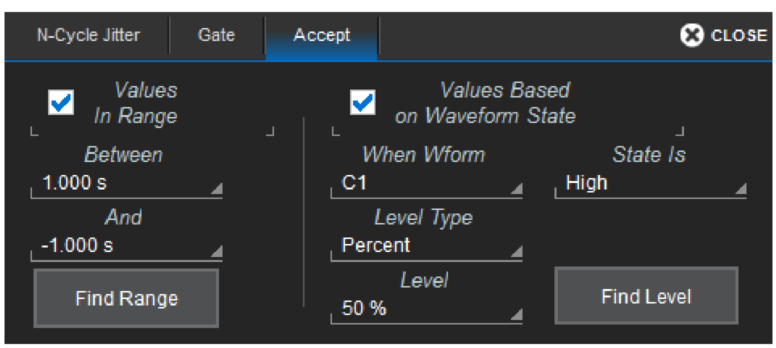


Accepting Measurements

Some measurements can be constrained to a vertically or horizontally limited range of values, or to samples "gated" by the state of a second waveform. Both constraints can operate together. This capability enables you to exclude unwanted characteristics from your measurements. It is much more restrictive than a measurement gate, which is used only to narrow the span of analysis along the horizontal axis.

Note: Since this feature operates on only a subset of the data, possible alerts or status indicators concerning the measurement (such as "Data range too low") are not displayed.

If the measurement supports this feature, you will see the Accept subdialog next to the P*n* dialog for you to define your constraints.



Range Limited Parameters

1. From the menu bar, choose **Measure > Measure Setup...**, then touch the **P*n*** tab to open the parameter setup dialog.
2. On the **Accept** subdialog, select **Values In Range** and enter the start and stop values, or touch the **Find Range** button to quickly display the most recently measured range of values.

> Note: The correct units for the measurement (V, s, Hz, dB, etc.) are automatically displayed. If you select a simple ratio parameter that yields a dimensionless number (such as power factor), units are omitted.

Waveform Gated Parameters

1. From the menu bar, choose **Measure > Measure Setup**, then touch the **P*n*** tab to open the parameter setup dialog.
2. On the **Accept** subdialog, select **Values Based on Waveform State**.
3. Touch **When Wform** and select the gating waveform. It can be any waveform active at the same time as the measurement source waveform.
4. Touch **State Is** and select **High** or **Low** from the pop-up menu. Measurements will only be taken when the gating waveform is in the selected state.
5. Touch **Level Type** and select **Absolute** (voltage) or **Percent** (of amplitude) from the pop-up menu.
6. Enter the crossing **Level** value at which you want measurements to begin.

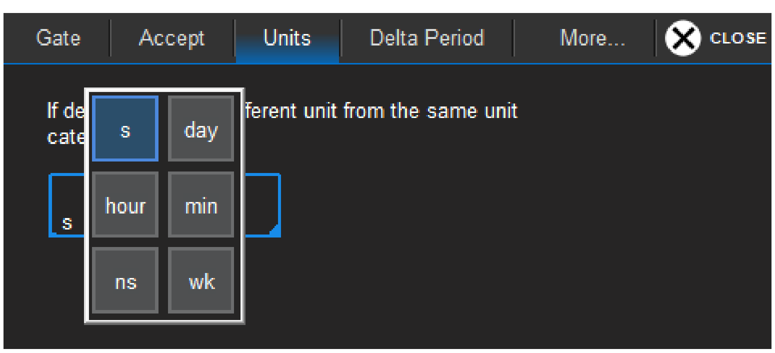
   You can also touch the **Find Level** button to automatically set the level at 50% of the gating waveform.

Changing Units

Use the Units subdialog to change the output unit of a measurement parameter or math function trace. The dialog is only active when there are alternative units in the equivalent dimension as the source unit.

Unit Category is the unit type, for example, Length, Velocity, or Time.

Units reflect the Category selection, for example, a Length unit of meters (m) versus a Velocity unit of meters per second (m/s).



List of Standard Measurements

Note: Unless otherwise stated, measurements are calculated according to IEEE standards. Additional measurements may be available depending on the software options installed. The keyword to use in remote control programs appears in Summary on the P*n* dialog.

Waveform Measurements

Amplitude

Difference between the upper and lower levels in two-level (bi-modal) signals. Differs from Peak-to-Peak (pkpk) in that noise, overshoot, undershoot, and ringing do not affect the measurement. Amplitude is calculated using the formula for Top-Base. On signals that cannot be identified as bi-modal (e.g., triangle waveforms), Amplitude returns the same value as Maximum – Minimum. Amplitude may be calculated once per period, rather than once per acquisition, by selecting "Show one value per period" on the Amplitude subdialog.

Area

Integral of data. Computes the area of the waveform relative to the zero level. Values greater than zero contribute positively to the area; values less than zero contribute negatively. If Cyclic is checked on the Area subdialog, the area is calculated only over the total number of full cycles, rather than the entire acquisition. See Periodic Measurements.

Base

Lower level in two-level (bi-modal) signals (the higher is Top), or lower of two most probable waveform states on waveforms that are not bi-modal. Base differs from Minimum in that noise, overshoot, undershoot and ringing do not affect the measurement. On signals that are not bi-modal (e.g., triangle waveforms), Base returns the same value as Minimum. Base may be calculated once per period, rather than once per acquisition, by selecting "Show one value per period" on the Base subdialog.

Bit Rate

Bit rate of input data stream, calculated on rising edges only, so it is not affected by DCD. The measurement must be made on at least three rising edges, so that there is more than one interval between rising edges. Otherwise, the measurement returns the clock frequency, and a warning symbol will appear on the Measure table.

Cycles

Number of cycles in a periodic waveform that occur between the measurement gates (by default, 0 and 10 divisions, or the full acquisition). The first cycle begins at the first rising threshold crossing after the left measurement gate.

Delay

Time from the acquisition trigger to the first 50% level crossing visible in the observation window. On acquisitions without a Timebase Delay setting, this is usually a negative number.

Delta Delay

Time from the first 50% level crossing of Source1 to the first 50% level crossing of Source2. Delta Delay can be used to measure the propagation delay between two signals by triggering on one and measuring the Delay of the other. It may yield a negative result in cases where the Source2 crossing occurs before the Source1 crossing. See Gating Measurements.

Dperiod@level

Cycle-to-cycle deviation of the period measurement, measured from rising edges (Pos Slope), falling edges (Neg Slope), or next crossing (Both Slope) at the specified Level. See Slope, Level and Hysteresis. By default, it measures the adjacent cycle deviation (cycle-to-cycle jitter) for each cycle in a waveform, but it may be configured to compare cycles at set intervals or the mean value of groups of cycles by using an N-Cycle Setup. See Periodic Measurements.

Dtime@level

Time between transitions on two, different input signals, measured on rising edges (Pos Slope), falling edges (Neg Slope), or next crossing (Both Slope) at the specified Level. Dtime@level may yield a negative result in cases where the Source2 crossing occurs before the Source1 crossing. See Gating Measurements and Slope, Level and Hysteresis.

Dtrig Time

Time from last acquisition trigger to the present acquisition trigger. For Sequence Mode acquisitions, the time between segment triggers within a single segmented waveform.

Duration

Time from first acquisition trigger to last acquisition trigger included in an average, histogram, or segmented waveform. For single-sweep acquisitions, Duration is 0. For segmented waveforms (Sequence mode acquisitions), Duration is the time from the first segment trigger to the last segment trigger (that occurs between the measurement gates). For averages, histograms and other waveforms calculated over a history of acquisitions, Duration is the time from the first to the last accumulated waveform's trigger.

Duty Cycle

Percent of period for which data are above or below the 50% level of the signal, using a hysteresis band of 22% of amplitude.

Duty@level

Percent of period for which data are above or below a specified level, measured on rising edges (Pos Slope) or falling edges (Neg Slope). See Slope, Level and Hysteresis.

Dwidth@level

Difference between adjacent Widths, measured on rising edges (Pos Slope), falling edges (Neg Slope), or next crossing (Both Slope) at the specified Level.

Edge to Edge

Time from the specified edge on Source1 to the specified edge on Source2.

Edge@level

Number of edges in waveform that cross the specified threshold Level, measured on rising edges (Pos Slope), falling edges (Neg Slope), or next crossing (Both Slope). See Slope, Level and Hysteresis.

Fall 80-20%

Duration of a pulse waveform's falling transition from 80% to 20% of the amplitude, averaged for all falling transitions between the measurement gates. On signals that do not have two major levels (e.g., triangle waveforms), the Top-Base measurement used to calculate Amplitude can default to Maximum and Minimum, giving less predictable results.

Fall Time

Duration of a pulse waveform's falling transition from 90% to 10% of the Amplitude, averaged for all falling transitions between the measurement gates. On signals that do not have two major levels (e.g., triangle waveforms), the Top-Base measurement used to calculate Amplitude can default to Maximum and Minimum, giving less predictable results.

Fall@level

Duration of pulse waveform's falling transition between two, user-specified transition levels, averaged for all falling transitions between the measurement gates. Two vertical values on the falling edge are used to compute fall time:

High = upper threshhold value x Amplitude/100 + Base
Low = lower threshold value x Amplitude/100 + Base

See Slope, Level and Hysteresis.

First

Time at which the left measurement gate intersects the Horizontal axis, marking the first sample point included in the measurement. By default, this is the time at 0 divisions, set at the far left of the grid.

Frequency

Reciprocal of each Period of cyclic signal. Period is measured as time between every pair of 50% crossings on the rising edge, starting with the first rising transition after the left measurement gate.

Freq@level

Reciprocal of each Period of a cyclic signal. Period is measured as the time between every pair of crossings at the specified level and edge, starting with first matching transition after left measurement gate. See Slope, Level and Hysteresis.

Half Period

Half period of a waveform, measured as the time between adjacent transitions of opposite slope at a specified level. You can configure this measurement to be taken from only rising edges to falling edges

(Pos Slope), from only falling edges to rising edges (Neg Slope), or from every crossing to the next (Both Slope). See Slope, Level and Hysteresis.

Hold Time

Time from the first data signal edge at specified slope and level (Source2), to nearest previous clock signal edge at specified slope and level (Source1). Use Hold Clock subdialog to specify clock edge, and Hold Data subdialog to specify the data edge.

Last

Time at which the right measurement gate intersects the Horizontal axis, marking the last sample point included in the measurement. By default, this is the time at 10 divisions, set at the far right of the grid.

Level@X

Vertical value at the specified x-axis position, which may be set by entering a value or moving the white measurement cursor. If the position is between two sample points, it gives the interpolated value. When the Nearest point checkbox is selected, it gives the vertical value of the nearest data point.

Maximum

Largest vertical value in a waveform. Unlike Top, does not assume the waveform has two levels.

Mean

Average of vertical values in a waveform. Computed as centroid of distribution for a histogram of the data values.

Median

Average of Top and Base values.

Minimum

Smallest vertical value in a waveform. Unlike Base, does not assume the waveform has two levels.

N-Cycle Jitter

Peak-to-peak jitter between edges measured at *N* UI intervals. Results may be viewed in time (S) or UI.

NBPhase

Narrow Band Phase, phase at a specified frequency of the signal.

NBPower

Narrow Band Power, power at a specified frequency of the signal.

Num Points

Number of data points in the source waveform between the measurement gates.

Overshoot-

Amount of overshoot following falling edges, represented as percentage of amplitude. Overshoot- is calculated using the formula (Base - Minimum)/Amplitude x 100. On signals that do not have two major levels (e.g., triange waveforms), this measurement may not give predictable results.

Overshoot+

Amount of overshoot following rising edges, represented as a percentage of amplitude. Overshoot+ is calculated using the formula (Maximum - Top)/Amplitude x 100. On signals that do not have two major levels (e.g., triangle waveforms), this measurement may not give predictable results.

Peak to Peak

The difference between the maximum and minimum vertical values within the measurement gates. Unlike Amplitude, does not assume a waveform has two levels.

Period

The time between 50% crossings on the rising edge, starting with the first transition after the left measurement gate. Period is measured for each adjacent pair, with values averaged to give the final result.

Period@level

The time between crossings at a user-specified slope and level, starting with first transition after the left measurement gate. See Slope, Level and Hysteresis. By default, Period is measured for each adjacent pair, with values averaged to give the final result, but it can be configured to compare cycles at set intervals by using an N-Period Setup. See Periodic Measurements.

Phase

Phase difference between analyzed and reference signals, measured from the 50% level of their rising edges.

Rise 20-80%

Duration of a pulse waveform's rising transition from 20% to 80% of amplitude, averaged for all rising transitions between the measurement gates. On signals that do not have two major levels (such as triangle or saw-tooth waves), the Top-Base measurement used to calculate rise can default to Maximum and Minimum, giving less predictable results.

Rise Time

Duration of a pulse waveform's rising transition from 10% to 90% of amplitude, averaged for all rising transitions between the measurement gates. On signals that do not have two major levels (e.g., triangle waveforms), the Top-Base measurement used to calculate rise can default to Maximum and Minimum, giving less predictable results.

Rise@level

Duration of a pulse waveform's rising edge between user-defined transition levels, averaged for all rising edges between the measurement gates. The two values used to compute the rise time are calculated as:

Low = lower threshold x Amplitude/100 + Base
High = upper threshold x Amplitude/100 + Base

See Slope, Level and Hysteresis.

RMS

Root Mean Square of the vertical values (between the measurement gates), calculated using the formula:

$$\sqrt{\frac{1}{N}\sum_{i=1}^{N}(V_i)^2}$$

Where: V_i = measured vertical values, and N = number of data points.

If Cyclic is checked on the RMS subdialog, the RMS is calculated over only full cycles, rather than the entire acquisition. See Periodic Measurements.

Setup

Time from data edge (Source2) to next nearest clock edge (Source1).

Skew

Time of Clock2 edge (Source2) minus the time of previous Clock1 edge (Source1).

Slew Rate

Slew rate (local dV/dt) at user-specified slope and level. See Slope, Level and Hysteresis.

Std Dev

Standard deviation of the vertical values between the measurement gates, using the formula:

$$\sqrt{\frac{1}{N}\sum_{i=1}^{N}(V_i - mean)^2}$$

Where: V_i = measured vertical values, and N = number of data points. This is equivalent to the RMS for a zero-mean waveform. Also referred to as AC RMS.

If Cyclic is checked on the Std Dev subdialog, the standard deviation is calculated over only full cycles, rather than the entire acquisition. See Periodic Measurements.

TIE@level

Difference between the measured crossing time at the specified slope and level and the ideal expected time. The measurement may be given in time (S) or unit interval (UI). A UI equals one clock period. The Virtual Clock (set up on the VClock subdialog) enables you to simulate your choice of Standard (1.544 MHz) or Custom reference clocks. You can also use a mathematically derived Golden PLL to filter low frequency jitter. The cutoff frequency is user selectable. See Slope, Level and Hysteresis.

Time@level

Time from trigger (t=0) to crossing at a specified slope and level. See Slope, Level and Hysteresis.

Top

Higher vertical value in two-level (bi-modal) signals (the lower is Base), or higher of two most probable waveform states on waveforms that are not bi-modal. Top differs from Maximum in that noise, overshoot, undershoot and ringing do not affect the measurement. On signals that do not have two major levels (e.g., triangle waveforms), Top returns the same value as Maximum. Top may be calculated once per period, rather than once per acquisition, by selecting "Show one value per period" on the Top subdialog.

Width

Width of cyclic signal measured at 50% level and positive slope, using a hysteresis of 22% of amplitude. Widths of all waveform pulses are averaged for the final result.

WidthN

Width of cyclic signal measured at 50% level and negative slope, using a hysteresis of 22% of amplitude. Widths of all waveform pulses are averaged for the final result.

Width@level

Width of cyclic signal measured at a user-specified level, slope and hysteresis. Widths of all waveform pulses are averaged for the final result. See Slope, Level and Hysteresis.

X@max

The horizontal axis location of the maximum vertical value between the measurement gates.

X@min

The horizontal axis location of the minimum vertical value between the measurement gates.

Histogram Measurements

These measurements can be made on histogram math functions. You will receive an error status if you attempt to place them on a pulse waveform or other incompatible source.

FWXX

Full Width at xx%, width of the largest area histogram peak at a user-specified percentage of the population of the highest peak.

Hist ampl

Difference in value of the two most populated peaks in a histogram when the distribution is bi-modal. If the distribution is not bi-modal, it is the difference between the Maximum and Minimum values in the histogram.

Hist base

Value of the left-most of the two most populated histogram peaks.

Hist max pop

Count of the maximum population (highest) peak in a histogram.

Hist maximum

Highest measured value in a histogram.

Hist mean

Average or mean value of measurements in a histogram.

Hist median

Median value of measurements in a histogram, the 'X-axis' of a histogram that divides the population into two equal halves.

Hist minimum

Lowest measured value in a histogram.

Hist mode

Value represented by the highest histogram peak, shown in the unit of the parameter that is being histogrammed.

Hist pop@x

Population of the histogram bin at a specified horizontal position.It is helpful to Stop Trigger and turn on Axis Labels for this measurement so you can see the visible range of horizontal values. Once you enter a value in this range, the histogram cursor will appear for you to drag to different positions along the X axis. The value reported on the Measure table will change as you drag the cursor.

Hist range

The Max-to-Min range of a histogram.

Hist rms

Root mean square of the values in a histogram.

Hist sdev

Standard deviation of the values in a histogram.

Hist top

Value of the right-most histogram peak.

Hist X@peak

Value of the *n*th highest histogram peak. You supply the value of *n*.

Peaks

Number of peaks in a histogram.

Percentile

Horizontal data value that divides a histogram so that the population to the left is xx% of the total.

Total Pop

Total population (number of values plotted) of a histogram.

Custom Measurements

These "measurements" enable you to insert a custom script into the oscilloscope process and read back results into the parameter in real time.The script must yield a numeric result.

Custom measurements are standard on WaveMaster, LabMaster, DDA and SDA models. They are enabled with the XDEV option on others.

ExcelParam

Custom measurement calculated in Excel. The measurement will transfer one or two waveforms (as a series of time-value pairs) into an Excel spreadsheet and read the result of the function into the parameter.

Fast MultiWave Port

Provides memory mapped C++ file access using multiple inputs for custom measurement calculation. The result may optionally be read back into the parameter.

HParamScript

Custom Visual Basic or Java script that calculates a measurement on one or two input histograms, as opposed to waveforms.

MathCad param

Custom MathCad function that calculates a measurement.

MATLAB param

Custom MATLAB function that calculates a measurement. You may write a MATLAB script on the oscilloscope, or call a pre-existing MATLAB function into a script. MATLAB can run locally on the oscilloscope, or remotely on a host that is accessible to the oscilloscope. See the *MAUI Oscilloscopes Remote Control and Automation Manual* for instructions on connecting to MATLAB remotely.

ParamScript

Visual Basic or Java script that calculates a measurement from one or two input waveforms. The script may run locally on the oscilloscope, or remotely on a host that is accessible to the oscilloscope. See the *MAUI Oscilloscopes Remote Control and Automation Manual* for instructions on connecting remotely.

Parameter Math

In addition to measuring waveforms, you can set up a parameter that performs mathematical operations on the results of other parameters. Parameter math differs from math functions in that the input and the output are still numerical values that display in the Measure table. Math functions, on the other hand, input and output waveform traces that appear on the grid.

The setup for parameter math is much like that for other parameters. Some parameters can be qualified on the Accept subdialog using value ranges or gating waveforms, as can regular waveform measurements. If you have the Advanced Customization (XDEV) option, there is functionality for applying custom scripts to calculate the results.

Exclusions

The following are not supported by Parameter Math:

- Multiplication and division of parameters that return logarithmic values
- Source parameters that are the result of other parameter math operations

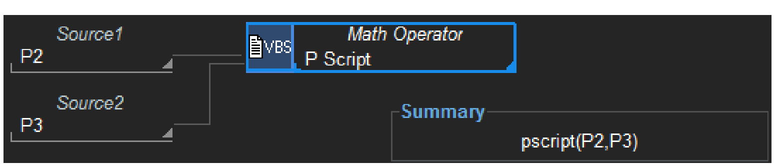
Setting Up Parameter Math

1. From the menu bar, choose **Measure > Measure Setup...**.
2. Choose Measure Mode **My Measure** and open a parameter setup (P*n*) dialog.
3. Touch the **Math on Parameters** button.
4. Touch **Math Operator** and choose an operation from the **Select Measurement** menu.
5. Touch the **Source** fields and select the parameters that are the inputs to the measurement.
6. Check **On** to enable the new output parameter and add it to the measurement readout.

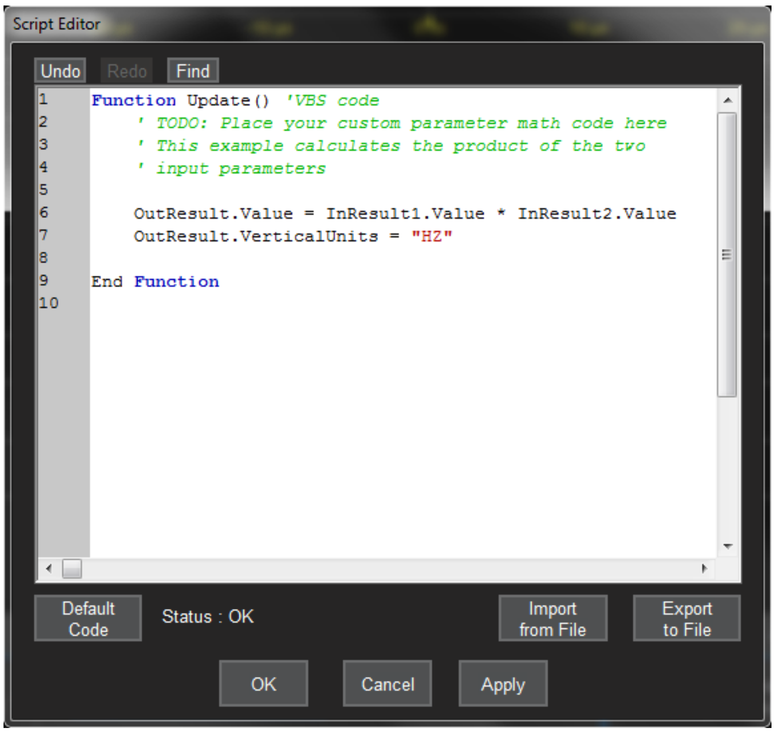
Using P Script (Custom Parameter Math)

On instruments installed with the Advanced Customization option (XDEV), you can write your own VBScript or JavaScript to apply a mathematical operation to one or two input parameters. Choose the Math Operator **P Script**. Scripting can be done directly on the instrument in the Script Editor window, or you can import an existing script.

Note: Param Script is a VBScript or JavaScript that performs a measurement on one or two *waveforms* to output a parameter. P Script, which is Parameter Math, is a VBScript or JavaScript that inputs one or two *parameter values* and performs a math operation on them to output another parameter.



1. Choose **Measure > Measure Setup** from the menu bar.
2. Touch the **P*n* tab** of the output parameter to display the parameter setup dialog.
3. Touch the **Math on Parameters** button.
4. Touch **Math Operator** and choose **P Script**.
5. Enter the **Source** input parameter(s).
6. On the Script Math subdialog, touch **Script Language** and choose either VBScript or JScript.
7. Touch the **Edit Code** button to open the Script Editor window.



8. Enter your code in the window, or **Import from File** an existing script. Click **Apply** and **OK** to finish.

Scripts created in this window can be exported to a new file for future use.

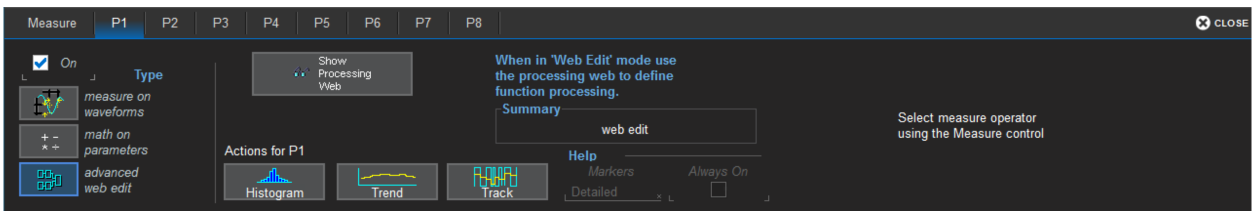
Using Web Editor

Whereas with Parameter Math you can apply a single math operation to measurement parameters, the Web Editor enables you to create processes that chain many operations on multiple inputs, including custom math and measurements (with the XDEV option), and yield multiple outputs. These processes are integral to the operation of the instrument; there is no need to export data to other programs.

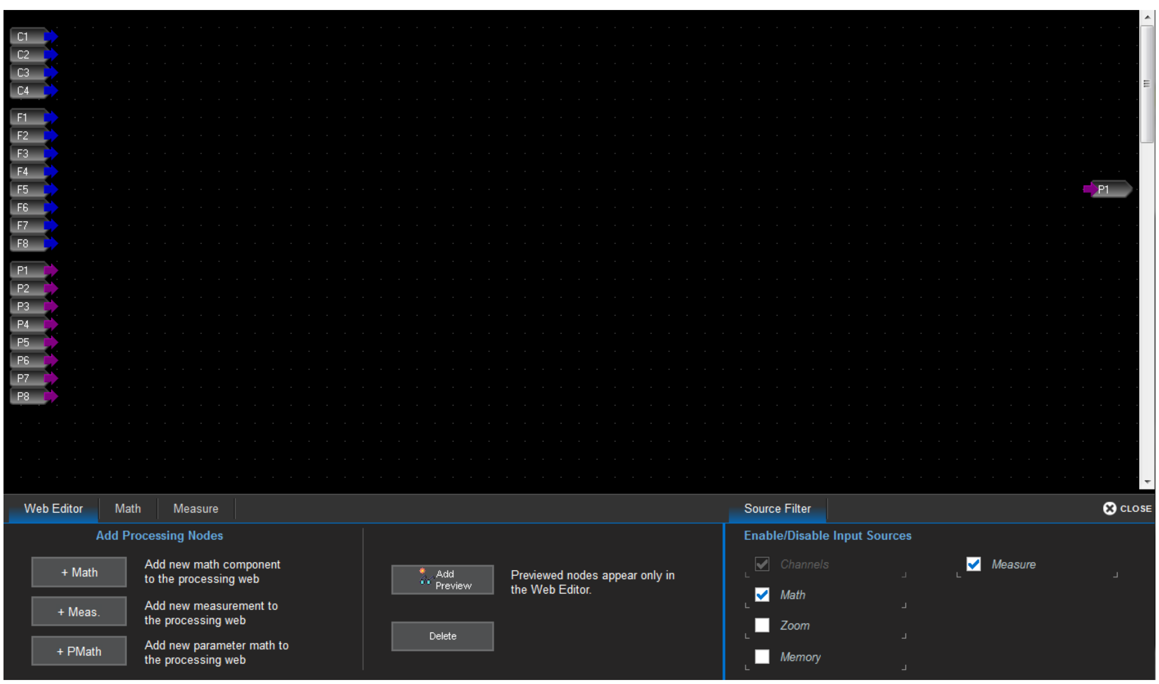
Processes are "programmed" in the X-Stream application simply by arranging a series of blocks representing different math functions or measurements and connecting them to form a flow chart—a "processing web." The terminal parameters and functions represent the final outputs of this process.

Create Processing Web

1. From the menu bar, choose **Measure > Measure Setup**.
2. On the Measure dialog, choose Measure Mode **My Measure**.
3. Open the **P*n*** dialog and select **Advanced Web Edit**.



4. Touch the **Show Processing Web** button. You will see a "peg board" display with potential input sources aligned along the left, and a terminal marked with the parameter number to the far right.



5. On the Source Filter subdialog, deselect any types (e.g., zooms) that you do not want to use in the process. This simplifes the display and reduces the need to scroll.

6. On the Wed Editor dialog, choose the type of process to add to the flow. You can **Add Math**, **Add Measure**, or **Add Param Math**.

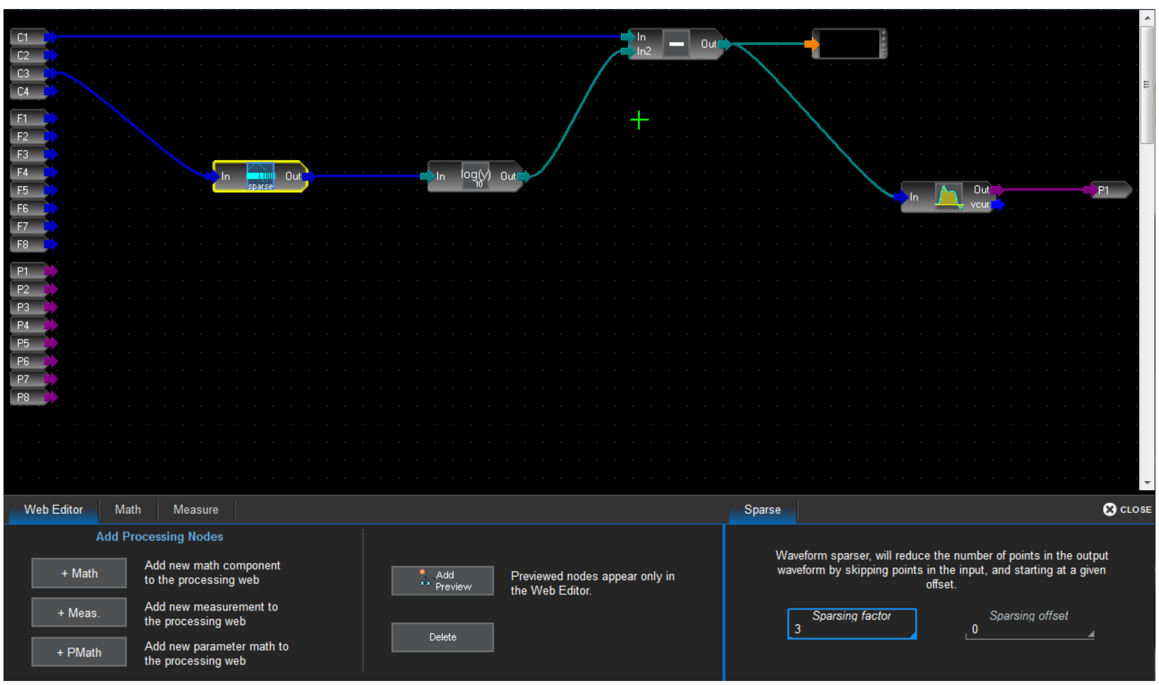
7. From the pop-up, choose the operation or measurement that will occur at that node. A new block appears on the display. Drag the block to a location in the flow.

> **Tip:** Touch the location to place a green plus sign before adding the node. Then, the new block will be created at this spot. Otherwise, blocks may cover one another until moved.

8. The process block will have a input "pin" on the left. If a waveform is required, the pin is blue; if a number is required, the pin is purple. Touch and drag a matching colored pin from the sources on the left of the screen to the block. Choose as many sources as there are input pins on the block.

> **Tip:** You may need to use the scrollbar to see all the available sources. For convenience, the Math and Measure dialogs behind the Web Editor dialog summarize what is currently configured for those sources.

When you drop the pin, a line is drawn from the source to the process. If a source is incompatible with a process, you will not be able to pin it.



9. If the process requires additional configuration, a subdialog appears next to the Web Editor dialog. Use it to enter the values to apply to that processing node.

10. Continue to add processes as needed, pinning the output of each block to the input of the next block in the flow. The final process should be the same type as the terminal (in this example, a measurement).

 > **Tip:** The color of the input/output arrows indicates which connections are compatible.The flow does not need to be strictly sequential, as many processes can accept inputs that have undergone other processes. The only requirement is that the outputs are pinned to acceptable inputs.

11. Drag the output pin from the final process block to the input pin on the terminal.

Preview Outputs

You can add preview "windows" to see the output of a process at any stage in the flow. These previews apply only to the Web Editor.

Select the output pin of the processing block you wish to preview. It will turn bright green. Then, on the Web Editor dialog choose **Add Preview**.

Delete Process/Connection

To delete any process from the web, select the block then touch **Delete**. All the connections to/from it are automatically deleted.

To remove a connection between process blocks, touch the line. A scissor icon appears above it. Confirm whether or not to "cut" the connection.

Add Other Terminals to Processing Web

Other measurements or math functions can share the same processing web. For example, you may wish to create a math function of the waveform that would result from a sub-processing node, while the final output of the full process is a measurement parameter.

To add a terminal:

1. Open the Math or Measure dialog behind the Web Editor dialog.
2. Touch the Web Edit icon following the location (P*n* or F*n*). The summary changes to Web Edit, and a new terminal block appears on the Web Editor peg board.
3. Connect the output pin of the desired process to the input pin of the terminal. You can select outputs that are already pinned to other blocks.

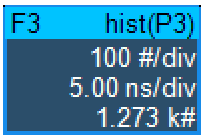
Exit Web Editor

When your processing web is complete, close the Web Editor dialog or choose Scope Display from the Display menu. The screen will resume the regular oscilloscope display.

Using Histograms

Histograms display the statistical distribution of parameter values using bars of varying heights to represent measurement frequency. The Histogram math function plots the number of parameter measurements that fall within a small range, known as a bin, as a function of the parameter value. Each bin is represented by a bar on the chart. The histogram can use from 20 to 5000 bins, each representing a consecutive sub-range of measured values, and it can hold up to two billion measured values.

The Phistogram function creates a histogram of parameter values falling within a vertical or horizontal "slice" of a persistence map (the trace created when Persistence display is turned on during acquisition).



The Histogram function descriptor box shows: the number of values represented by each vertical grid division of the histogram bars (#/div), the bin width represented by each bar of the histogram (vertical or horizontal units/div), and the total number of values computed in the histogram (#).

Thumbnail versions of parameter histograms are called Histicons. They are available as a checkbox option on the Measure dialog. Histicons appear on the Measure table, rather than as a new math trace, although they can be converted to a regular histogram function simply by touching them.

Uses of Histograms

Histograms show the distribution and probability density of waveform parameter values over multiple acquisitions. They are especially useful for visualizing and quantifying random processes, such as jitter. The shape of the histogram yields immediate information about the parameter being measured:

Gaussian/Unimodal histograms have one peak near the center of the distribution with roughly equilateral roll off on each side. The Gaussian distribution is a good indication that a random process is shaping variations in the measurement, such as the effect of random jitter on a pulse measurement.

Bimodal histograms have two, distinct peaks in the distribution at the outer edges, indicative of steady-state high and low values.

Uniform/Symmetric histograms look rectangular, indicating a wide distribution of values.

Skewed histograms show a peak at either the negative or positive edge of the measurement range, with a steady roll off toward the other side.

Because the shape of histogram distributions is particularly interesting, additional histogram measurements are available for analyzing these distributions. They are generally centered on one of several peak value bins. You will find the histogram measurements in the Statistics sub-menu of the Measure Selector.

Comparing the total populations of each bin of the histogram can help quantify rarely occurring events that might otherwise be overlooked, one advantage of the statistical study of measured data. Example applications of histograms include:

- Characterizing DDR read/write timing limits
- Examing systemic and random variations that produce jitter and time interval error

- Measuring rise time, setup / hold time, and propagation delay of electronic devices
- Detecting and diagnosing circuit problems, such as crossover distortion

Histogram Calculation

Histograms are calculated on the portion of the waveform within the acquisition window and the measurement gates, just as are the parameters. Likewise, histogram measurements are limited to the bars of the histogram that are visible on the grid *and* that fall within the measurement gates.

In Sequence sampling mode, one acquisition for each segment occurs prior to parameter calculations, and a sweep is equivalent to acquisitions for all segments (for non-segmented waveforms, an acquisition is identical to a sweep). If a single segment of a sequence-mode acquisition is selected for viewing, histograms will be recalculated for only the measurements on that segment.

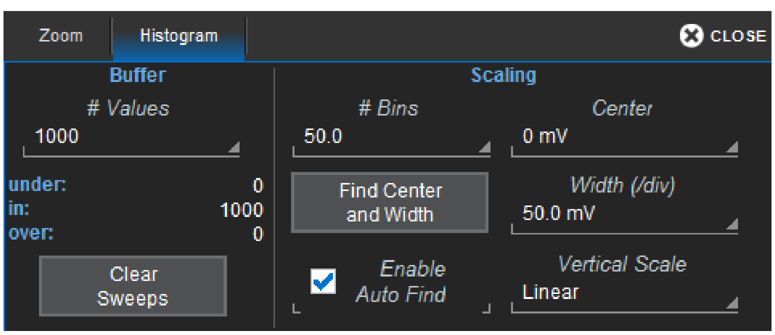
Measurements that yield only one value per acquisition (Amplitude, Area, Base, Cycles, Delay, Delta Delay, Duration, First, Last, Maximum, Mean, Median, Minimum, Nb Phase, Nb Power, Overshoot+, Overshoot-, Peak-Peak, Phase, Points, RMS, Std Dev., etc.) will not produce characteristic histogram distributions, unless you allow sweeps to accumulate.

Plotting Histograms

1. Open the parameter setup dialog (P*n*) for the parameter you wish to histogram.

   Tip: A quick way to do this is to touch the Measure table P*n* cell.

2. Touch the **Histogram** action button at the bottom of the P*n* dialog and choose the function (**F*n***) in which to draw the histogram.
3. Touch the new **hist descriptor box** to display the F*n* dialog, then open the **Histogram subdialog** and define the histogram Buffer and Scaling.



Pressing **Clear Sweeps** at any time resets the measurement counter and restarts the histogram.

Histogram Buffer

Enter the maximum **#Values** in one bin of the histogram. This determines the total number of measurements that are represented by a bar at full height. Once this number is reached, the histogram will be rescaled according to the Vertical Scale method selected, as it continues to accumulate data.

Histograms are dynamically computed using the last *N* measurements in the parameter buffer, up to two billion. If #Values is modified, the histogram is redrawn using the last *N* events required.

If the parameter source is a memory, storing new data to memory effectively acts as an acquisition trigger/sweep and updates the parameter buffer and the histogram.

Histogram Scaling

Bins

First, enter the total #Bins that comprise the histogram. This determines how many bars appear in the histogram.

Determining peak value bins, or histogram peaks, is very useful because they indicate the dominant values of a parameter. However, the use of a high number of bins relative to the number of parameter values acquired can yield a spiky histogram, making meaningful peaks hard to distinguish.

- The default value of 100 is a good number for Vertical (amplitude) parameters.
- For Horizontal (timing) parameters, it is safe to use up to the maximum number of bins.

An optimal bin number is one where the change in bin value is insignificant, and the histogram distribution does not have a jagged appearance. The instrument analyzes histogram data to identify peaks from background noise and histogram definition artifacts such as small gaps, which are due to very narrow bins.

Histogram bins represent a sub-range of waveform parameter measurements. The measurements that fall within a bin may have a value anywhere within its sub-range. However, *measurements of the histogram itself* assume that all measurements in a bin have a single value. The instrument uses the center value of each bin's sub-range in all its histogram measurements. The greater the number of bins used to subdivide a histogram's range, the less the potential deviation between actual measured values and those values assumed in histogram measurements. Nevertheless, using more bins may require a greater number of sweeps to populate the bins sufficiently for the identification of a characteristic histogram distribution.

Center and Width

How parameter values are distributed into the bins is determined by the Center and Width values.

The Center is the center value of the measurement range represented by the histogram. This will generally be a value near the expected peak value, or a center bar in a symmetric histogram. It is a good idea to examine the mean value of the parameter you wish to histogram to determine the value to place at center.

The Width is the measurement interval represented by each bin of the histogram, which is counted in vertical units/div for Vertical parameters, or horizontal units/div for Horizontal parameters. If Width is 500 ns, a Horizontal measurement that is less than or equal to 500 ns from center falls into the first positive bin, and one that is less than or equal to -500 ns from center falls into the first negative bin, and so forth for each increment of 500 ns and successive bins. The more bins that are in the histogram, the smaller the Width should be to ensure that values will distribute across the histogram, showing a meaningful deviation from center.

When the ideal range required to form a good distribution is not apparent, the instrument's range finding function, Find Center and Width, can be used to calculate an optimal range and redraw the histogram using it. The instrument gives a running count of where values fall relative to the range on the Histogram function subdialog, and if any fall above or below it, the range finder can recalculate the histogram using those values while still in the buffer. First check Enable Auto Find to use this feature.

Vertical Scale

Histograms build vertically as data accumulates over multiple acquisitions. It is possible that, after a sufficient number of sweeps, the peak bars of a histogram will reach the grid maximum, causing an overflow condition, and the histogram must be rescaled to accurately reflect the distribution. Choose a Vertical Scale method by which to handle situations where the histogram buffer is full:

- Linear rescales the number of values represented by each vertical division as the histogram reaches the top of the grid. The histogram will appear to shrink then grow again as more data is accumulated, until another rescaling of the values/div is required.
- LinConstMax keeps the histogram at near full scale when it reaches the top of the grid, but changes the numbers on the vertical axis as more data is accumulated.

Using Trends

The Trend math function plots a waveform composed of parameter measurements arranged in the order the measurements were made. The vertical units are the source parameter values, and the horizontal unit is the measurement number. The Trend contains a single value for each unique measurement, and therefore may not be time synchronous with the source waveform, where the same measured value may occur successively over time.

Uses of Trends

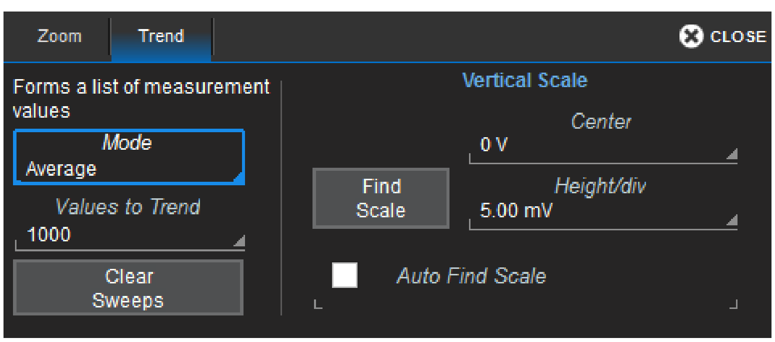
Trends are especially useful for visualizing the history of a parameter over an extended period of time or over multiple acquisitions. Think of Trend as a strip chart recorder for your instrument. Example applications of Trend include:

- Data logging multiple circuit parameters
- Power line monitoring
- Measuring output regulation and ripple
- Characterizing the dynamic tracking response of PLL-based circuits
- Locating clock jitter anomalies

Plotting Trends

Although a Trend plots parameter values, it is created as a Math function on the Function (F*n*) dialogs.

1. Open the parameter setup (P*n*) dialog for the measurement you wish to trend. A quick way to do this is to touch the Measure table cell.
2. Touch the **Trend button** at the bottom of the Pn dialog, then choose the function (**F*n***) in which to draw the plot. The Trend opens in a new grid along with its function descriptor box.
3. Touch the new **Trend descriptor box** to display the F*n* dialog, then open the **Trend subdialog**.



4. Choose a computation **Mode** of All (measurements per acquisition) or Average (one measurement per acquisition).
5. Enter the number of measured **Values to Trend**.

Using Tracks

A Track is a waveform composed of parameter measurements that are time synchronous with the source waveform. The vertical units are those of the source parameter and the horizontal units are seconds. Due to the time synchronization, the number of points in the Track function is identical to the number of samples in the source waveform, and a Track is limited to the samples in a single acquisition. The Track may contain many redundant values.

Uses of Tracks

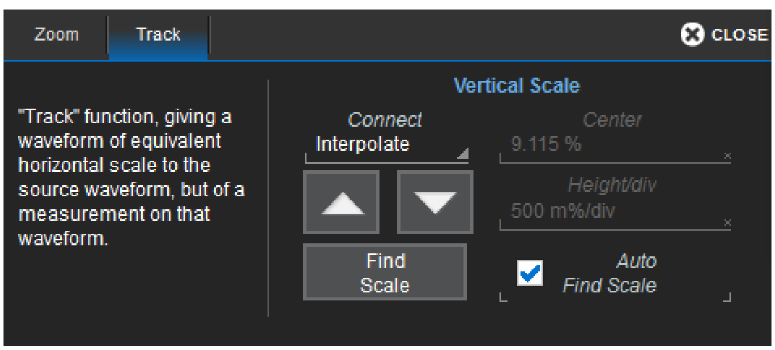
In general, Track is the tool to use if you want to capture a continuous stream of data spaced closely together. It is also useful in applications that require time synchronization, or for locating anomalous parameter values in a long acquisition. Example applications of Track include:

- Modulation analysis of PWM signals
- Measuring phase error in PLLs
- Measuring settling time of waveforms with decaying amplitudes
- Radar measurements to characterize PRI stagger during walkoff

Plotting Tracks

Although a Track plots parameter values, it is created as a Math function and controlled on the Function (F*n*) dialogs.

1. Open the parameter setup (P*n*) dialog for the measurement you wish to track. A quick way to do this is to touch the Measure table cell.
2. Touch the **Track** toolbar button at the bottom of the P*n* dialog and choose the function (**F*n***) in which to draw the plot.
3. Touch the new **Track descriptor box** to display the F*n* dialog, then open the **Track subdialog**.



4. On the Track subdialog, choose a method to **Connect** the points in the track into a waveform: Interpolate points between samples, or Extend the line directly point to point.

Track vs. Trend

The table below summarizes the differences between Track and Trend.

	Track (Parameter value vs. time)	**Trend (Parameter value vs. event)**
Cumulative?	No, resets after every acquisition	Cumulative over unlimited number of acquisitions, up to two billion events
Time correlates to other data?	Yes	No
Monitors Frequency domain?	Yes	No, points are not evenly spaced in time and therefore cannot be used for an FFT
Monitors parameters over multiple acquisitions?	No, resets after every acquisition	Yes
Preserves all measurement data?	Yes, although maximum time period that can be captured is limited by acquisition memory and sampling rate	No, data can be missed during time oscilloscope takes to re-arm between acquisitions

EMC Pulse Parameter Software Package

The EMC option enhances the oscilloscope's Measure functionality with features designed to ignore undershoot, overshoot, or tail perturbations while measuring rise time, fall time, or width characteristics according to specific EMC/ESD standards.

At Level Parameters

For the parameters Rise@Level, Fall@Level, Width@Level, Time@Level, and dTime@level, in addition to setting the crossing threshold level using the absolute voltage or percentage of amplitude based on the computed Top and Base lines, you may set the level using:

- actual Peak to Peak
- 0V - Max
- 0V - Min

Once the option is activated, the new selections are added to the Level pop-up menu. Otherwise, parameters are configured exactly the same as before.

EMC Parameters

Two parameters designed for EMC measurements are added to the IEC-EMC submenu of the Measure Selector:

- EMClvlPulse—EMC Level After Pulse enables you to measure the signal level following the specified pulse transition, omitting earlier portions of the signal.
- EMCt2Val—EMC Time to Half Value computes time between interpolated time at 0% on rising edge and crossing time at middle level on falling edge. The interpolated time at 0% is obtained by linear extrapolation using the low crossing and high crossing thresholds.

Math

Math function traces (F*n*) display the result of applying a mathematical operation to a source trace. The output of a math function is always another trace, whereas the output of a measurement parameter is a tabular readout of the measurement.

Math can be applied to any channel (C*n*), zoom (Z*n*), or memory (M*n*) trace. It can even be applied to another math trace, allowing you to chain operations (for example, trace F1 can show the average of C1, while trace F2 provides the integral of F1).

On instruments equipped with CustomDSO (XDEV), custom scripts may be used to define a math function. Several scripting "languages" are supported, depending on the application. You can find these options in the Custom submenu of the Math Selector. Scripts may be imported or written in the instrument's Script Editor window and saved for future use. See Using P Script for a description of the process.

In addition to the extensive math capabilities that are standard with every instrument, enhanced math analysis tools customized for various industries and applications are offered through optional software packages. To learn about math tools available in each optional package, see the product datasheets at teledynelecroy.com.

If you have installed software options, the new capabilities are usually accessed through the Analysis menu, rather than the Math menu, although special math functions will be available when using the standard Math dialogs.

Note: If there is a processing error (e.g., overflow) when calculating a math function, a small letter "i" inside a bubble will appear on the F*n* descriptor box to indicate there is more information regarding the waveform status. See Finding Waveform Status for instructions on finding the error.

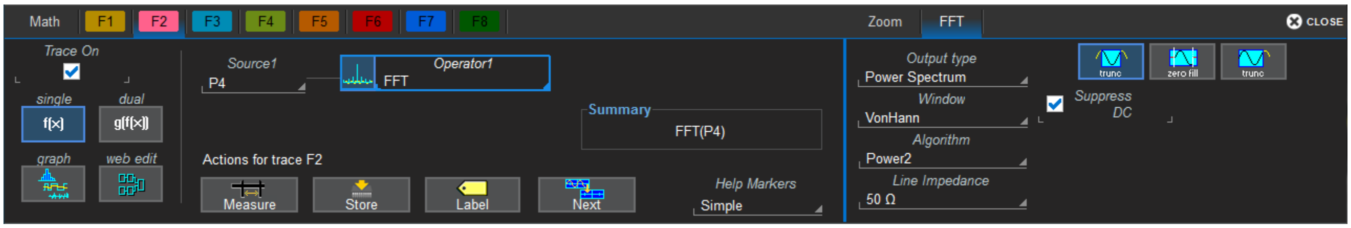
Math Function Set Up

Use the Function dialog to set up math function traces. Math functions take as input one or more channel, zoom, memory or math traces and output a new math trace (F*n*). Any additional settings required for the operator will appear on a subdialog at the right of the screen.

Single functions perform one operation on one or two input sources.

Dual functions chain two operations to arrive at a single result. This saves you the effort of having to chain two separate math functions. As with single functions, the number of sources required will vary based on the operation. You may need only one source for Operator1, but two for Operator2 (the result of the first operation counts as one source).

Setting Up New Functions



1. From the menu bar choose **Math > Math Setup**, then open one of the **F*n* tabs**.

   Tip: You can select **F*n* Setup** right from the Math menu.

2. Choose a **single f(x)** or **dual g(f(x)** operator function.

   You can also choose Web Edit, in which case this function is added as a terminal to the processing web. See the instructions for Using Wed Edit to set up the function.

3. In **Operator1**, choose the math operation to perform.

   Note: On instruments with the Advanced Customization (XDEV) option, the Operator can be a custom script written in one of several scripting "languages." Choose from the Custom submenu and import or write the script in the Script Editor. See Using PScript for a description of the process.

4. The choice of operator drives the number of **Source** fields you will see displayed. Make a selection in each field, or drag the source channel descriptor box to the field.

   A **Summary** of the function you are building appears on the dialog. Refer to this to be sure your sources are in the proper order to yield the function you want (e.g., C1-C2 vs. C2-C1).

5. If the operator you've selected has any other configurable settings, you'll see a subdialog of the same name as the operator. Touch the tab to open the dialog and make any further settings. These are explained on the dialog.

6. If you're creating a dual function, repeat the procedure for the second operator.

Graphing

The **Graph button** on the Function (F*n*) dialogs allows you to choose a parameter to plot as a histogram, track, or trend.

Tip: The plots are the same as those created using the toolbar on the Parameter (P*n*) dialog.

As with other math functions, configurable settings will appear on subdialogs after the plot is selected.

Adjusting Memory or Math Traces

Unlike channel traces, the scale of memory (M*n*) or math function (F*n*) traces can be adjusted directly without having to create a separate zoom trace. The same set of zoom factor controls used for zoom traces appear on the **Zoom subdialog**, but in this context they only rescale the active math or memory trace rather than create a new zoom. This applies to any trace that is created as a math function (F*n*) trace, including traces generated through analysis options and graphs.

You can, if you wish, create a separate zoom trace from a memory or function trace the same as you would normally create a zoom (draw a selection box, etc.). In this case, you choose one of the zoom locations (Z*n*) in which to draw the trace, but the source trace remains at the original scale.

List of Standard Math Operators

Note: The installation of software options on the oscilloscope may add math operators to this list.

Standard Operators

Absolute

Calculates distance away from zero for every point in the waveform. For values greater than zero, this is the same as the value. For values less than zero, the magnitude without regard to its sign is used.

Average

Calculates either a summed or continuous average of a selected number of sweeps. See Averaging Waveforms. The maximum number of sweeps is determined by the oscilloscope model and memory.

Copy

Copies waveform in its unprocessed state to the first available memory location.

Correlation

Calculates a measure of similarity of two waveforms, or a waveform against itself, as a function of a time-lag applied to one of them.

Derivative

Calculates the derivative of adjacent samples using the formula:

(next sample value – current sample value) / (horizontal sample interval)

Deskew

Shifts trace in time the amount of the deskew factor.

Difference

For every point in the waveform, subtracts the value of Source2 from the value of Source1. Source1 and Source2 must have the same horizontal and vertical units and scale.

Envelope

Calculates highest and lowest vertical values of a waveform at each horizontal value for a specified number of sweeps.

ERes

Applies a noise reduction and smoothing filter by adding a specified number of bits. See Enhanced Resolution.

Exp

Calculates the antilog to the base e of the source; that is, e raised to the power equal to the source.

Exp10

Same as Exp, using base 10.

FFT

Computes a frequency spectrum with optional Rectangular, Von Hann, Flat Topp, Hamming, Blackman-Harris, and Hanning windows. Calculates up to 128 Mpts. Also allows FFT Averaging through use of a second math operator. See FFT.

Floor

Calculates the lowest vertical values of a waveform at each horizontal value for a specified number of sweeps.

Histogram

Plots the number of data points that fall into statistically significant intervals or bins. Bar height relates to the frequency at which data points fall into each interval/bin.

Integral

Calculates the linearly rescaled integral (with multiplier and adder) of a waveform input starting from the left edge of the screen using the formula:

*(current sample value + next sample value) * (horizontal sample interval)*

Each calculated area is summed with the previous sum of areas. The multiplier and adder are applied before the integration function.

Interpolate

Inserts points between sampled points (upsamples) according to one of three algorithms: Linear (straight line), Sinx/x (curved), and Cubic (spine). The Interpolation Factor determines the number of points in the upsample.

Invert

For every point in the waveform, the inverse of that point is calculated.

Ln

Peforms a natural log of a waveform. Values less than or equal to zero are set to underflow.

Log10

Performs a log base 10 of a waveform. Values less than or equal to zero are set to underflow.

Phistogram

Creates a histogram based on the displayed pixels of a persistence map falling within a user-defined vertical and/or horizontal box (slice). The source trace must have Persistence turned "on."

Product

For every point in the waveform, the value of Source1 is multiplied by the value of Source 2. Source1 and Source2 must have the same horizontal units and scale.

Ptrace mean

Plots the mean value of each sample point in a persistence map. The source trace must have Persistence turned "on."

Ptrace range

Generates a waveform with a width derived from the population range of a persistence map. The source trace must have Persistence turned "on."

Ptrace sigma

Generates a waveform with a width derived from the sigma (sum) of a persistence map. The source trace must have Persistence turned "on."

Ratio

For every point in the waveform, divides the value of Source1 by the value of Source2. Source1 and Source2 must have the same horizontal units and scale.

Reciprocal

For every point in the waveform, calculates the inverse using the formula: 1 / (sample value).

Rescale

For every point in the waveform, multiplies the sample value by the specified Multiplier, then adds the specified Additive Constant value. See Rescaling and Assigning Units.

Roof

Calculates the highest vertical value at each sample point for a specified number of sweeps.

Segment

Selects one segment [#] from a segmented waveform and copies it into the function, enabling it to be rescaled independent of the source trace.

Sinx/x

Performs 10-to-1 interpolation using a Sin(x)/x filter.

Sparse

"Thins," or decimates, an incoming acquisition by dropping sample points at regular intervals. Sparsing Factor specifies the number of points to drop between retained samples (e.g., factor of 4 retains 1 then drops 4). Sparsing Offset specifies the point at which to begin applying the sparsing factor (e.g., offset of 3 begins count on the third sample (3), then drops the number of samples specified by the sparsing factor (4).

Square

For every point in the waveform, calculates the square of the sample value.

Square Root

For every point in the waveform, calculates the square root of the sample value.

Sum

For every point in the waveform, adds the value of Source1 to the value of Source 2. Source1 and Source2 must have the same horizontal and vertical units and scale.

Track

Generates a waveform composed of measurement parameter values that are time synchronous with the source waveform. The vertical units are those of the source parameter value; the horizontal units are seconds. Values are posted at the sampling rate.

Trend

Produces a waveform composed of a series of measurement parameter values in the order the measurements were taken. The vertical units are those of the source parameter; the horizontal unit is measurement number. The trend contains a single value for each unique measurement.

Zoom

Produces a magnified trace of a selected portion of the input waveform. See Zooming Traces.

Custom Math Operators

These "operators" enable you to insert a custom script into the oscilloscope process and view the resulting waveform in real time.

Excel math

Custom math performed in Excel. The operator will transfer one or two waveforms (as a series of time-value pairs) into an Excel spreadsheet and display the resulting waveform.

Fast Wave Port

Provides memory mapped C++ file access for custom math calculations.

Math script

Visual Basic or Java script that produces a waveform from one or two input waveforms.

Mathcad math

Produces a waveform using a custom Mathcad function.

MATLAB math

Produces a waveform using a custom MATLAB function. You may write a MATLAB script on the oscilloscope, or call a pre-existing MATLAB function into a script. MATLAB can run locally on the oscilloscope, or remotely on a host that is accessible to the oscilloscope. See the *MAUI Oscilloscopes Remote Control and Automation Manual* for instructions on connecting to MATLAB remotely.

Average Function

The summed or continuous average of all data samples from multiple acquisitions can be displayed as a new waveform trace using the Average function.

Setting Up Averaging

To apply Continuous or Summed Averaging as a Math function:

1. Follow the usual steps to set up a math fuction, selecting **Average** from the **Basic Math** submenu.
2. On the **Average** subdialog, choose **Summed** or **Continuous**.
3. Touch **Sweeps** and provide a value.

Tip: To quickly set up Continuous Averaging (only), access the channel setup dialog (C*n*) and enter the number of sweeps to average in Averaging.

Summed Averaging

Summed Averaging is the repeated addition, with equal weight, of successive source waveform records. If a stable trigger is available, the resulting average has a random noise component lower than that of a single-shot record. Whenever the maximum number of sweeps is reached, the averaging process stops. In Summed averaging, you specify the number of acquisitions to be averaged. The averaged data is updated at regular intervals.

An even larger number of records can be accumulated simply by changing the number in the dialog. However, the other parameters must be left unchanged or a new averaging calculation will be started. You can pause the averaging by changing the trigger mode from Normal/Autoto Stop. The instrument resumes averaging when you change the trigger mode back to Normal/Auto.

You can reset the accumulated average by pushing the Clear Sweeps button or by changing an acquisition parameter such as input gain, offset, coupling, trigger condition, timebase, or bandwidth limit. The number of current averaged waveforms of the function, or its zoom, is shown in the acquisition status dialog. When summed averaging is performed, the display is updated at a reduced rate to increase the averaging speed (points and events per second).

Continuous Averaging

Continuous Averaging, the default setting, is the repeated addition, with unequal weight, of successive source waveforms. It is particularly useful for reducing noise on signals that drift very slowly in time or amplitude. The most recently acquired waveform has more weight than all the previously acquired ones: the continuous average is dominated by the statistical fluctuations of the most recently acquired waveform. The weight of 'old' waveforms in the continuous average tends to zero (following an exponential rule) at a rate that decreases as the weight increases.

You determine the importance of new data vs. old data by assigning a weighting factor. The formula for continuous averaging is:

$$\text{new average} = (\text{new data} + \text{weight} * \text{old average})/(\text{weight} + 1)$$

By setting a **Sweeps** value, you establish a fixed weight that is assigned to the old average once the number of sweeps is reached. For example, for a sweeps (weight) value of **4**:

Sweep	New Average =
1 (no old average yet)	(new data +0 * old average)/(0 + 1) = new data only
2	(new data + 1*old average)/(1 + 1) = 1/2 new data +1/2 old average
3	(new data + 2 * old average)/(2 + 1) = 1/3 new data + 2/3 old average
4	(new data + 3 * old average)/(3 + 1) = 1/4 new data + 3/4 old average
5	(new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average
6	(new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average
7	(new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

Note: The number of sweeps used to compute the average is displayed at the bottom of the trace descriptor box.

Copy Function

The **Copy** math function saves a copy of your present waveform in its unprocessed state to the first available memory location. The memory enables faster throughput, in some cases, by preserving the original data, while math and measure processing continues on the original waveform. This benefit of faster throughput, however, comes at the expense of memory usage.

Follow the ususal steps to set up a math function, selecting **Copy** from the **Misc** submenu.

On the Wform Copy subdialog, optionally **Reset Count** or **Change BatchSize**.

ERes Function

ERes (Enhanced Resolution) filtering increases vertical resolution, allowing you to distinguish closely spaced voltage levels. The instrument's ERes function is similar to smoothing the signal with a simple, moving-average filter, but is more efficient concerning bandwidth and pass-band filtering. Use ERes:

- On single acquisitions or where the data is slowly repetitive (and you cannot use averaging).
- To reduce noise on noticeably noisy signals when you do not need to perform noise measurements.
- As a low-pass filter. The ERes filter rejects high-frequency components from the signal. The higher the bit enhancement, the lower the resulting bandwidth.
- When performing high-precision voltage measurements (e.g., zooming with high vertical gain).

Setting Up ERes

To apply ERes as a Math function:

1. Follow the usual steps to set up a math function, selecting **Eres** from the **Filter** submenu.
2. Touch the **Trace On** checkbox.
3. On the **Eres** subdialog, select the number of **bits** of improvement from the pop-up menu.

Tip: To quickly set up ERes as a pre-processing function, access the channel setup dialog (C*n*) and select a **Noise Filter (ERes)** bit size.

How ERes Is Applied

The instrument's ERes feature improves vertical resolution by a fixed amount for each filter. This real increase in resolution occurs whether or not the signal is noisy, or whether it is single-shot or repetitive. The signal-to-noise ratio (SNR) improvement depends on the form of the noise in the original signal. ERes filtering decreases the bandwidth of the signal, filtering out some of the noise.

The instrument's constant phase finite impulse response (FIR) filters provide fast computation, excellent step response in 0.5 bit steps, and minimum bandwidth reduction for resolution improvements of between 0.5 and 3 bits. Each step corresponds to a bandwidth reduction factor of two, allowing easy control of the bandwidth resolution trade-off.

Resolution Increase	-3 dB Bandwidth (x Nyquist)	Filter Length (Samples)
0.5	0.5	2
1.0	0.241	5
1.5	0.121	10
2.0	0.058	24
2.5	0.029	51
3.0	0.016	117

With low-pass filters, the actual SNR increase obtained in any particular situation depends on the power spectral density of the noise on the signal.

The improvement in SNR corresponds to the improvement in resolution if the noise in the signal is white (evenly distributed across the frequency spectrum). If the noise power is biased towards high frequencies, the SNR improvement will be better than the resolution improvement.

The opposite may be true if the noise is mostly at lower frequencies. SNR improvement due to the removal of coherent noise signals—feed-through of clock signals, for example—is determined by the fall of the dominant frequency components of the signal in the passband. This is easily ascertained using spectral analysis. The filters have a precisely constant zero-phase response. This has two benefits. First, the filters do not distort the relative position of different events in the waveform, even if the events' frequency content is different. Second, because the waveforms are stored, the delay normally associated with filtering (between the input and output waveforms) can be exactly compensated during the computation of the filtered waveform.

The filters have been given exact unity gain at low frequency. ERes should therefore not cause overflow if the source data is not overflowed. If part of the source trace were to overflow, filtering would be allowed, but the results in the vicinity of the overflowed data—the filter impulse response length—would be incorrect. This is because in some circumstances an overflow may be a spike of only one or two samples, and the energy in this spike may not be enough to significantly affect the results. It would then be undesirable to disallow the whole trace.

Note: While ERes improves the resolution of a trace, it cannot improve the accuracy or linearity of the original quantization. The pass-band causes signal attenuation for signals near the cut-off frequency. The highest frequencies passed may be slightly attenuated. Perform the filtering on finite record lengths. Data is lost at the start and end of the waveform and the trace ends up slightly shorter after filtering. The number of samples lost is exactly equal to the length of the impulse response of the filter used: between 2 and 117 samples. Normally this loss (just 0.2 % of a 50,000 point trace) is not noticed. However, you might filter a record so short that no data is output. In that case, however, the instrument would not allow you to use the ERes feature.

Fast Wave Port Function

Fast Wave Port is an Advanced Customization (XDEV) function of Teledyne LeCroy X-Stream oscilloscopes that enables you to insert a custom algorithm written in the C/C++ language into the oscilloscope's processing stream. It also maximizes data throughput from the acquisition system to your custom algorithm for purposes of debugging or external processing. Custom algorithms can be written right on the oscilloscope using the Script Editor.

Because two Windows system processes may share a single region of memory, Fast Wave Port enables high-speed data transfer between the acquisition system and the custom algorithm, which runs in a separate process from the oscilloscope application. A major benefit of Fast Wave Port is that your algorithm may be implemented and, more importantly, debugged independently of the main application.

Transferring the results of your algorithm back into the X-StreamDSO processing stream is optional. If performance is the primary goal, and further processing or display of the results within the X-StreamDSO software is not required, this step may be skipped.

Fast Wave Port was designed for use with the C/C++ programming language. However, it is theoretically possible for the processing to be implemented in any language supporting Windows named events (Mutex) that can open a named memory-mapped file. No guarantee can be given, however, as to the behavior of the function using anything but C/C++.

The size of the memory window is fixed at 80 MB, equating to 40 ms.

Tip: See Fast Wave Port in the oscilloscope online Support for an example C++ application.

To use Fast Wave Port, follow the usual steps to set up a math function, selecting **Fast Wave Port** as the Operator, then make the appropriate settings on the **Fast Wave Port** subdialog.

The **Port Name** is critical, indicating the base name of the memory window and the global objects within the Windows O/S. Only change the default value if using multiple Fast Wave Port functions in parallel.

Note: The base name must match the base name used in the client application.

The full names of the default global objects are as follows:

Object	Full Name
Memory Mapped File	FastWavePort1File
Data Available Event	FastWavePort1MutexDataAvailable
Processing Complete Event	FastWavePort1MutexProcessingComplete

The **Timeout** control specifies the amount of time the DSO waits for the custom processing function to complete. This prevents the DSO from waiting indefinitely for a custom processing function which may never arrive. Be careful to set this control to a reasonable value, meaning a time longer by a reasonable margin than the custom processing is ever expected to take.

FFT Function

For a large class of signals, you can gain greater insight by looking at spectral representation rather than time description. Signals encountered in the frequency response of amplifiers, oscillator phase noise and those in mechanical vibration analysis, for example, are easier to observe in the frequency domain.

If sampling is done at a rate fast enough to faithfully approximate the original waveform (usually five times the highest frequency component in the signal), the resulting discrete data series will uniquely describe the analog signal. This is of particular value when dealing with transient signals, which conventional swept spectrum analyzers cannot handle.

While FFT has become a popular analysis tool, some care must be taken with it. In most instances, incorrect positioning of the signal within the display grid will significantly alter the spectrum, producing effects such as leakage and aliasing that distort the spectrum.

An effective way to reduce these effects is to maximize the acquisition record length. Record length directly conditions the effective sampling rate and therefore determines the frequency resolution and span at which spectral analysis can be carried out.

Setting Up FFT

1. Follow the usual steps to set up a math function, selecting **FFT** from the **Frequency Analysis** submenu.
2. Open the **FFT** subdialog.
3. Choose an **Output type**.
4. If your Output Type is Power Density or Power Spectrum, also enter **Line Impedence**. By default, the FFT function assumes a termination of 50 Ohms. If an external terminator is being used, this setting can be changed to properly calculate the FFT based on the new termination value.

   If your Output type is Phase, enter a **Group Delay Shift** value.
5. Optionally, choose a weighting **Window** (see below).
6. Check the **Suppress DC** box to make the DC bin go to zero. Otherwise, leave it unchecked.

Choosing a Window

If you think of an FFT as synthesizing a bank of parallel band-pass filters, weighting functions control the filter response shape and affect noise bandwidth as well as side lobe levels. Ideally, the main lobe should be as narrow and flat as possible to effectively discriminate all spectral components, while all side lobes should be infinitely attenuated. The window type defines the bandwidth and shape of the equivalent filter to be used in the FFT processing.

The choice of a spectral window is dictated by the signal's characteristics. Rectangular windows provide the highest frequency resolution and are useful for estimating the type of harmonics present in the signal. Because the rectangular window decays as a (SinX)/X function in the spectral domain, slight attenuation will be induced. Functions with less attenuation (Flat Top and Blackman-Harris) provide maximum amplitude at the expense of frequency resolution, whereas Hamming and Von Hann are good for general purpose use with continuous waveforms.

Window Type	Applications and Limitations
Rectangular	Normally used when the signal is transient (completely contained in the time-domain window) or known to have a fundamental frequency component that is an integer multiple of the fundamental frequency of the window. Signals other than these types will show varying amounts of spectral leakage and scallop loss, which can be corrected by selecting another type of window.
Hanning (Von Hann) & Hamming	Reduces leakage and improves amplitude accuracy. However, frequency resolution is also reduced.
Flat Top	Provides excellent amplitude accuracy with moderate reduction of leakage, but with reduced frequency resolution.
Blackman-Harris	Reduces leakage to a minimum, but with reduced frequency resolution.

FFT Window Filter Parameters				
Window Type	Highest Side Lobe (dB)	Scallop Loss (dB)	ENBW (bins)	Coherent Gain (dB)
Rectangular	-13	3.92	1.0	0.0
Von Hann	-32	1.42	1.5	-6.02
Hamming	-43	1.78	1.37	-5.35
Flat Top	-44	0.01	3.43	-11.05
Blackman-Harris	-67	1.13	1.71	-7.53

Interpolate Function

The Interpolate operator enables you to generate a trace that interpolates data points into the source trace according to your selected method and upsampling factor.

Linear interpolation, which inserts a straight line between sample points, is best used to reconstruct straight-edged signals such as square waves. This is the default interpolation method used by the oscilloscope.

(SinX)/X interpolation, on the other hand, is suitable for reconstructing curved or irregular waveshapes, especially when the sampling rate is 3 to 5 times the system bandwidth.

Cubic interpolation can be used to create a smooth, continuous function by applying a third-degree polynomial.

For each method, you can select an upsample factor of 2 to 50 points to insert between samples.

1. Follow the usual steps to set up a math function, selecting **Interpolate** from the **Filter** submenu.
2. Select an interpolation method.
3. Enter the **Upsample by** number of points.

> **Tip:** You can use the Up and Down buttons to set this value. To make single increment changes, deselect the "Only by 2, 5..." checkbox.

4. In **Half Width** enter the width in taps of the filter.
5. To use a weighting function rather than an upsample factor, select it from **Weighting**. See FFT for more information about these functions. If using the Kaiser function, also enter the **Beta** value.

Sparse Function

The Sparse math function allows you to thin out an incoming waveform by skipping points at regular intervals, and by starting acquisition at a particular offset (point). The **Sparsing factor** specifies the number of sample points to reduce the input waveform by. A sparsing factor of 4, for example, says to retain only one out of every 4 samples. A **Sparsing offset** of 3, on the other hand, says to begin on the third sample, then skip the number of samples specified by the sparsing factor (4). In this way, the sample rate is effectively reduced.

For the sparsing factor (interval), you can set a value from 1 to 1,000,000 points. For the sparsing offset you can set a value from 0 to 999,999.

1. Follow the usual steps to set up a math function, selecting **Sparse** from the **Misc** submenu.
2. Touch the **Sparsing factor** control and provide a Bandwidth Limit value.
3. Touch the **Sparsing offset** control and provide a value.

Rescale Function

The Rescale function allows you to create a new function trace that rescales another trace by applying a multiplication factor (*a*) and additive constant (*b*). You can also use it as a way to view the function source in a different unit of measure.

Tip: Channel and sensor traces may be rescaled directly using the C*n* and SE*n* dialogs.

Setting Up Rescaling

1. Follow the usual steps to set up a math function, selecting **Rescale** from the **Functions** submenu.
2. Touch the **Rescale** subdialog tab.
3. To modify the scale of output:
   - Check the **First multiply by:** box and enter the number of units equal to 1 V (*a*, the multiplication factor).
   - Touch **then add:** and enter *b*, the additive constant.
4. To change the output unit of measure from that of the source waveform:
   - Check **Override units**.
   - In **Output** enter the code for the new unit of measure.

   You can combine units following these rules:

   - For the quotient of two units, use the character "**/**"
   - For the product of two units, use the character "**.**"
   - For exponents, append the digit to the unit without a space (e.g., "S2" for seconds squared)

   Note: Some complex units are automatically converted to simple units. For example, `V.A` becomes `W`).

5. You can further override the output unit by selecting from the **Units** dialog. Only units in the equivalent dimension as the unit of the Rescale function trace are available for selection.

Units of Measure

Units are automatically rescaled up or down within the list of standard, SI prefixes based on the relative size of the signal. For example a 1000 V reading is shown as 1 kV, while .1 V is shown as 100 mV. When the multiplication factor is 1 V = 1 Pascal, a 10 millivolt (mV) reading is displayed as 10 mPa rather than .001 Pa or 100e-3 Pa.

Following are the supported SI units of measure and the mnemonics used to represent them on the Rescale dialog.

Note: These same mnemonics can be used in remote control programs and customization scripts. Specify only the base unit in code, do not add prefixes.

Note: Time and dimensionless units are available only for certain measurements and for use in code where relevant.

Category	Unit	Mnemonic
Mass	gram	G
	slug	SLUG
Volume	liter	L
	cubic meter	M3
	cubic inch	IN3
	cubic foot	FT3
	cubic yard	YARD3
Angle	radian	RAD
	arcdegree	DEG
	arcminute	MNT
	arcsecond	SEC
	cycle	CYCLE
	revolution	REV
	turn	TURN
Force/Weight	Newton	N
	grain	GR
	ounce	OZ
	pound	LB
Velocity	meters/second	M/S
	inches/second	IN/S
	feet/second	FT/S
	yards/second	YARD/S
	miles/second	MILE/S
Acceleration	meters/second^2	M/S2
	inches/second^2	IN/S2
	feet/second^2	FT/S2
	standard gravity	GN

Category	Unit	Mnemonic
Pressure	Pascal	PAL
	bar	BAR
	atmosphere, technical	AT
	atmosphere, standard	ATM
	Torr	TORR
	pounds/square inch	PSI
Temperature	degrees Kelvin	K
	degrees Celsius	CEL
	degrees Fahrenheit	FAR
Energy	Joule	J
	British Thermal Unit	BTU
	calorie	CAL
Rotating Machine	radians/second	RADPS
	frequency (Hertz)	HZ
	revolutions/second	RPS
	revolutions/minute	RPM
	torque N•m	NM
	torque in•oz	INOZ
	torque in•lb	INLB
	torque ft•lb	FTLB
	power, mechanical (Watt)	W
	horsepower	HP
Magnetic	Weber	WB
	Tesla	T
	inductance (Henry)	H
	magnetic field strength	A/M
	permeability	HENRYPM

Category	Unit	Mnemonic
Electrical	Ampere	A
	Volt	V
	Watt	W
	power, apparent	VA
	power, reactive	VAR
	power factor	PF
	capacitance (Farad)	F
	Coulomb	C
	Ohm	OHM
	Siemen	SIE
	electrical field strength	V/M
	electrical displacement field	CPM2
	permittivity	FARADPM
	conductivity	SIEPM
Time	second	S
	minute	MIN
	hour	HOUR
	day	DAY
	week	WEEK

Category	Unit	Mnemonic
Dimensionless	percent	PCT
	percent min-max	PCTMNMX
	decibel	DB
	decibel milliwatt	DBM
	decibel Volt	DBV
	decibel millivolt	DBMV
	decibel microvolt	DBUV
	decibel microampere	DBUA
	decibel referred to carrier	DBC
	decade	DECADE
	unit interval	UI
	Q-scale	Q
	bit	BIT
	byte	BYTE
	baud	BAUD
	least significant bit	LSB
	poise	POISE
	parts per million	PPM
	pixel	PIXEL
	division	DIV
	event	EVENT
	sample	SAMPLE
	segment	SEG
	sweep	SWEEP

Phistogram Function

The Phistogram function creates a histogram of the samples within a vertical or horizontal "slice" of a persistence map to reveal the features that are only visible when several acquisitions have been superimposed on one another. Vertically, each bin contains a class of similar amplitude levels; horizontally, each bin contains a class of similar time values.

The source trace must have Persistence "on" to generate the persistence map. Eye diagrams, which are a type of persistence map, are a an especially useful class of traces to which you can apply the Phistogram function.

For a histogram of...	Use...
A crossover point in time or amplitude on an eye diagram *	Vertical and horizontal slice persistence histogram
Cumulative jitter on an eye diagram *	Horizontal slice persistence histogram
Signal-to-noise ratio on an eye diagram *	Vertical slice persistence histogram

* Requires optional software package capable of creating eye diagrams, such as SDA III or Serial Decoder TDME software.

1. Follow the usual steps to set up a math function, choosing a **Single** operator function using the **Phistogram** operator.
2. On the **Phistogram** subdialog, choose the slice of the persistence map to histogram:
   - **Slice Direction** determines whether the slice transverses an entire vertical level of the map across time (horizontal), or a time on in the map across vertical values (vertical).
   - **Slice Center** represents the center level or time of the slice.
   - **Slice Width** represents the total amplitude or total time of the slice, half each side of the center. This setting will deepen or widen the total area of the slice of values to be plotted on the histogram. If Help Markers are on, you will see the change in the blue bar representing the slice.

   Note: This selection does not affect Center and Width settings made on any existing histograms.
3. Choose **Center Cursor** to place a measurement cursor at the center of the slice.

Ptrace Functions

The Ptrace Mean, Ptrace Range, and Ptrace Sigma math functions (found in the Jitter Functions sub-menu of the Math Operator selector) compute a vector trace based on a persistence map of underlying signal acquisitions without destroying the underlying raw data. Signals sampled at or above 2 GS/s that have accumulated in the persistence map can be traced at a resolution of 10 ps (100 GS/s equivalent sampling). Further measurements or Pass/Fail testing can be performed on the resulting math traces.

The source trace must have Persistence "on" to generate the persistence map used by each of these functions.

Ptrace Mean

The Ptrace Mean trace helps to see edge detail in a fast signal, or to eliminate noise on a persistence map.

For each time slice of the persistence map, Ptrace Mean calculates and plots a trace corresponding to the mean vertical value. The Ptrace Mean can be further analyzed using the instrument's standard parameters, such as rise time. Optionally, choose to limit the **Num**(ber of) **Points** plotted.

Ptrace Range

The Ptrace Range trace helps to assess typical noise on a persistence map.

For each time slice on the persistence map, Ptrace Range calculates and plots an envelope corresponding to the range of vertical values.

Enter the **% population range**, the percentage of the population of the persistence map that forms the envelope, enabling exclusion of infrequent events (artifacts). Optionally, choose to limit the **Num**(ber of) **Points** plotted.

Ptrace Sigma

The Ptrace Sigma trace helps to assess worst case noise on a persistence map. It can be used to create a tolerance mask for Pass/Fail testing.

For each time slice on the persistence map, Ptrace Sigma calculates and plots an envelope corresponding to the standard deviation of vertical values.

The envelope can be scaled to a specified sigma. Enter the **Scale to standard deviations**. This allows you to select a sigma from 0.5 to 10.0 to define the envelope of values plotted. Optionally, choose to limit the **Num**(ber of) **Points** plotted.

Math Dialog

Once a math function has been created and saved on the Function (F*n*) dialog, use the main Math dialog to quickly enable/disable it. You can also use this dialog to quickly turn on/off zoom traces.

To open the Math dialogs, from the menu bar choose **Math > Math Setup**. Select the **On** checkbox next to each function you wish to display.

To change the function, touch the **F*n* button**.

To erase all functions from their locations, touch **Reset All**.

To restart the counter on cumulative functions (like Average), touch **Clear Sweeps**.

Memory

The instrument is equipped with internal memory slots (M*n*) to which you can copy any waveform that is active on the grid. This is a convenient way to store an acquisition for later viewing and analysis. Memories can be used as source inputs for most oscilloscope math and measurements, allowing you to compare historical data to a live acquisition or perform "what if" modeling on saved acquisitions.

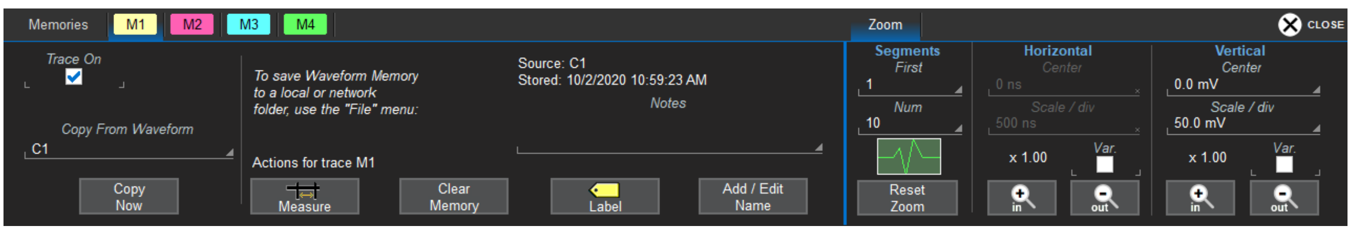
Note: If there is a processing error (e.g., no data) when saving or recalling a memory, a small letter "i" inside a bubble will appear on the M*n* descriptor box to indicate there is more information regarding the waveform status. See Finding Waveform Status for instructions on finding the error.

Saving Memories

Store memories on the Memory dialogs (M*n*). Memories are created at the same scale as the source trace, but they can be adjusted independently by using the zoom factor controls that appear next to the M*n* dialogs.

Save Waveform to Memory

Tip: Try to choose an empty slot, as anything currently stored in that location will be overwritten. All memories will state if they are empty or an acquisition is stored there.



On oscilloscopes with OneTouch, touch the **Add New box** and choose **Memory**. Drag the descriptor box of the trace you wish to store onto the M*n* descriptor box.

Or

1. Press the front panel **Mem button** or choose **Math > Memory Setup**.
2. Touch the **M*n* tab** corresponding to the memory slot you wish to use.
3. In **Copy from Waveform**, choose the source trace to copy to memory.
4. Touch **Copy Now**.
5. Optionally, check **Trace On** to immediately display the memory.

Save (External) Waveform Files to Memory

Trace (.trc) files saved on other Teledyne LeCroy instruments can also be imported into internal memory. Choose File > Recall Waveform and to save the file to an internal memory. Then, you can use the Memories dialog to restore them to the touch screen.

Restoring Memories

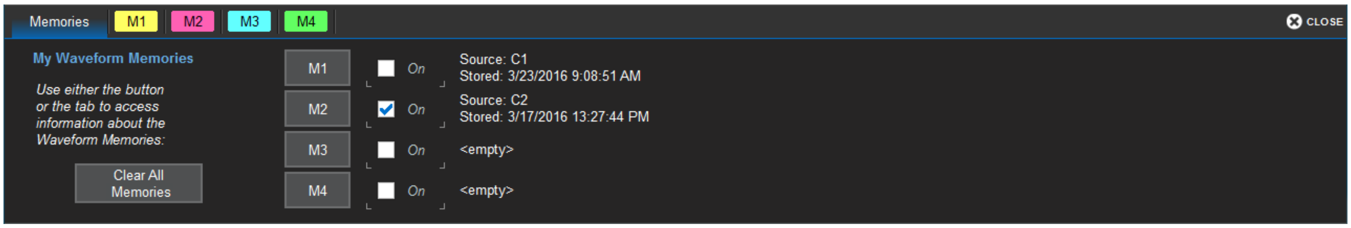
The Memories dialog is a convenient panel for restoring saved memories to the display.

Access the Memories dialog by pressing the front panel **Mem button** or choosing **Math > Memory Setup**.

Check **On** next to the memory trace you wish to display. A description of the memory showing the source channel and creation time appears next to each M*n* on the dialog.

Touch **Clear All Memories** to empty the memory banks.

Caution: Memories cannot be restored once they have been erased.



Analysis Tools

The Analysis menu tools complement the standard math and measurements to help you understand the behavior of waveforms.

WaveScan searches single or multiple acquisitions for events that meet specific criteria.

Pass/Fail Testing shows whether waveforms or measurements meet a set of defined criteria.

CustomDSO (XDEV) enables you to insert custom VBScript (VBS) or 3rd-party programs such as Excel™, Mathcad™, and MATLAB™ into the oscilloscope processing stream.

Optional software packages may be purchased for specialized uses, such as power analysis. In most cases, these options are added to the Analysis menu. Documentation for software options can be found at teledynelecroy.com/support/techlib under Manuals > Software Options. In addition, documentation for many options available on your platform can be found in the MAUI Support site under the Support menu.

WaveScan

The WaveScan® Search and Find tool enables you to search for unusual events in a single capture, or to scan for a particular event in many acquisitions over a long period of time. Each Scan Mode is optimized to find a different type of event. Results are time stamped, tabulated, and can be viewed individually.

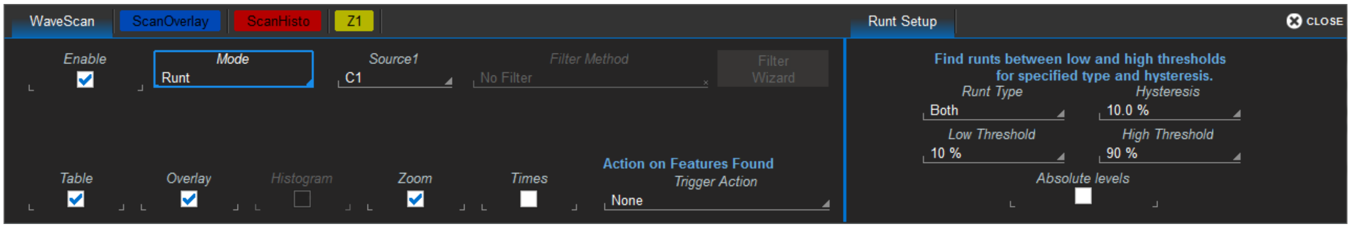
WaveScan window with different scan "views" turned on.

You customize the presentation by choosing different WaveScan displays, called Scan Views. Optionally, set Trigger Actions, such as stopping or beeping, to occur when the scanned events are found.

Note: The instrument reverts to Real-time sampling mode when WaveScan is enabled.

Setting Up WaveScan

Set up your source channel and triggers before setting up the scan.



1. Press the front panel **Stop button** to stop acquisition.
2. Choose **Analysis > WaveScan** and check **Enable**.
3. Choose the **Source** waveform.
4. Choose the Scan Mode and enter values for any additional settings that appear at the right of the dialog based on your selection.
5. Select each Scan View in which you wish to display results by checking the box at the bottom of the dialog. Each view selected is displayed simultaneously.
6. Optionally, choose a **Trigger Action** to take when an event is found that meets your scan criteria.

   Tip: Despite the name, these actions occur only when the WaveScan criteria are met, not with every acquisition trigger. Pulse AUX Output will send a pulse over the AUX Out connector. Print Screen will execute whatever you have configured on the Print dialog.

7. Restart acquisition.

Scan Modes

The Scan Mode determines the type of search to be performed. Select the **Mode** along with the **Source** trace to be searched on the main WaveScan dialog. For each mode, different controls appear on the WaveScan dialog, providing additional inputs to the search criteria. Make the appropriate entries in these fields before starting the search.

Edge Mode

Edge Mode is used for detecting the occurrence of edges. Events that meet the threshold level are captured and tabulated. When the acquisition is stopped, scan filters can be applied to the edges to find specific characteristics. Edge Mode settings are:

- **Slope**. Choose Pos, Neg, or Both.
- **Level is** (set in...). Choose Percent or Absolute.
- **Percent/Absolute Level**. Enter a threshold value as a percentage of Top to Base or voltage level.

Non-monotonic Mode

Non-monotonic Mode looks for edges that cross a threshold more than once between high and low levels. All events that meet the criteria of slope, hysteresis, and level are presented in a table and highlighted in the source trace. The value displayed in the table is the difference of the max. and min. of the non-monotonicity. This can be confirmed with cursors. The hysteresis value is used to eliminate noise. A non-monotonicity is detected only when its amplitude is greater than the hysteresis. Therefore, when setting a hysteresis level, set a value that is greater than the amplitude of the noise. Settings are:

- Slope. Choose Pos, Neg, or Both.
- Hysteresis is (set in...). Choose Division, Percent, Absolute.
- Division/Percent/Absolute. Enter hysteresis level in the selected unit.
- Levels are (set in...). Choose Percent, Absolute, or Pk-Pk%.
- High/Low Level. Enter top and bottom thresholds in the selected unit.

Runt Mode

Runt Mode looks for pulses that fail to cross a specified threshhold. You can search for positive-going or negative-going runts, or both. An adjustable hysteresis band is provided to eliminate noise.

In the case of negative-going runt pulses, the value displayed in the table is the difference (delta) of the high level of the signal and the runt amplitude (i.e., where the runt bottoms out). This can be confirmed by placing cursors on the runt pulse and reading the delta Y value in the trace labels. In the case of positive-going runt pulses, the value displayed in the table is the absolute value of the amplitude of the runt pulse. Runt Mode settings are:

- Runt Type. Choose Both, Pos, or Neg.
- Hysteresis. Enter the hysteresis level as a percentage or voltage.
- Low/High Threshold. Enter the levels as a percentage or voltage.
- Absolute Levels. Check this box to enter levels as absolute voltage instead of percentage.

Measurement Mode

Measurement Mode is used for applying filters to measurements to find those that meet your defined criteria, helping to isolate particular events within many samples. Markers appear over the source trace to indicate the location of measurement, while the table displays values for the selected parameter that meet the criteria. Measurement Mode settings are:

- Measurement. Choose the measurement parameter you wish to search.
- Filter Method. Choose the operator that indicates the desired relationship to the Filter Limit. Only measurements that meet this criteria are returned.
- Filter Limit. Enter the value that completes the filter criteria.

Alternatively, you can use the Filter Wizard to create the filter criteria.

Serial Pattern Mode

Serial Pattern Mode is used for finding 2- to 64-bit patterns in digital sequences. It is ideal for bursted patterns where a PLL cannot lock. Serial Pattern Mode settings are:

- **Viewing**. Choose to enter the pattern as Binary or Hex(adecimal).
- **Binary/Hex**. Enter the pattern.
- **Num. Patterns to detect**. Enter a whole number.

Bus Pattern Mode

Bus Pattern Mode (**only on Mixed Signal models**) is used for finding 2- to 16-bit patterns across the digital lines. Bus Pattern Mode settings are:

- **Viewing**. Choose to enter the pattern as Binary or Hex(adecimal).
- **Binary/Hex**. Enter the pattern.
- **Num. Patterns to detect**. Enter a whole number.

Scan Views

Scan Views are different ways to view your WaveScan results. Just check the boxes at the bottom of the WaveScan dialog for those views you wish to display simultaneously.

Note: The number of grids varies depending on which views are enabled. WaveScan handles this function automatically, and you cannot move traces among grids as in normal operation.

Additional controls for Scan Overlay and Zoom views are on their respective dialogs. If you turn on these traces from those dialogs, you must turn them off from there, too.

Source Trace

By default, the source trace is displayed in the top grid, with markers indicating points that meet the search criteria.

Table

Table view displays a table of measurements relevant to your chosen Search Mode next to the source trace. **Times** view adds columns to the table showing Start and Stop Times for each event.

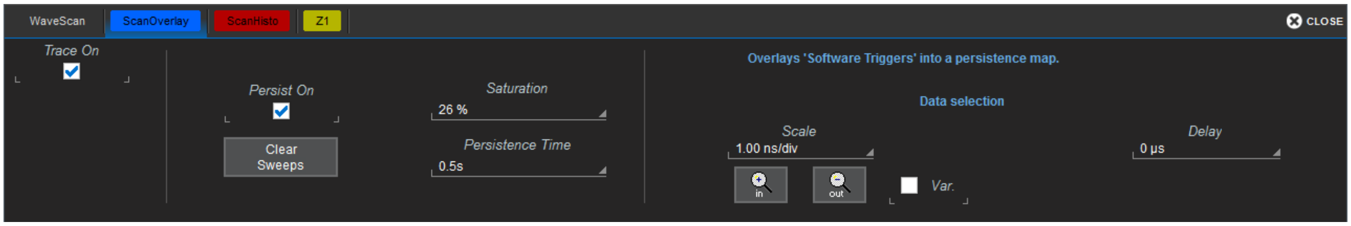
Scan Overlay

Scan Overlay view uses colored overlays to show the location of events meeting the criteria.

Zoom

Zoom view works exactly as it does elsewhere in the oscilloscope software, creating a new trace that is a magnified section of the source trace. A WScanZ*n* tab appears by default when you launch WaveScan; see zoom factor controls for an explanation of the remainder of the controls found on this dialog.

Scan Overlay



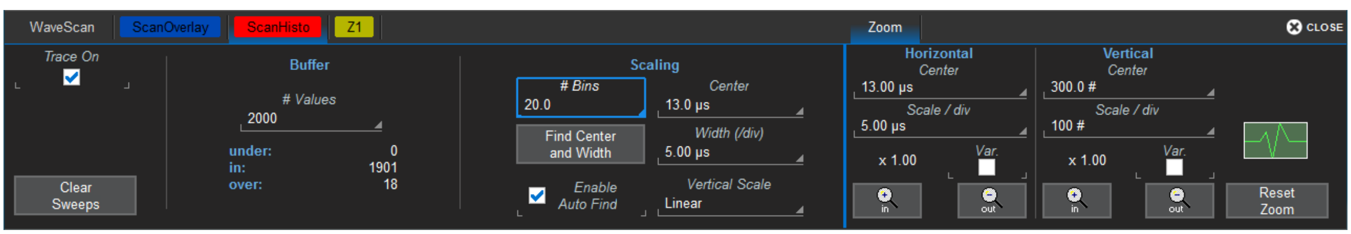
To apply monochromatic persistence to a Scan Overlay:

1. Select **Scan Overlay** when setting up the wave scan, then open the **Scan Overlay dialog**.
2. Check **Persistence On**.
3. Enter a **Saturation** level as a percentage. All samples above the saturation level are assigned the highest color intensity.
4. Choose a **Persistence Time**. The higher the time, the more static the persistence display.

To adjust the scan overlay to "zoom" in or out: touch the **In/Out buttons**, or touch **Scale** and enter new values. Check **Var.** to adjust scale in finer steps than the default 1, 2, 5.

Scan Histogram

Scan Histogram is an additional WaveScan "view" that generates a histogram to give you a statistical view of edges that meet your search criteria. Set it up as you would any histogram:



1. Enter the total **#Values** in the histogram buffer.
2. Enter the **#Bins** (bars) in the histogram.
3. Enter the **Center** value and **Width** of each bin from center, or check **Enable Auto Find** and let the sofware **Find Center and Width**.
4. Choose a **Vertical Scale** method:
   - Linear allows the histogram to build vertically as data accumulates. When the histogram reaches the top of the display it rescales the vertical axis to keep it on screen.
   - LinConstMax keeps the histogram at near full scale and rescales the vertical axis as data is accumulated.

WaveScan Search

Search is used to find events in traces—usually zoom (Z*n*) traces—that match user-defined criteria. To search within WaveScan:

1. Select the Zoom view.
2. After stopping the acquisition, open the Z1 dialog that appears behind the WaveScan dialog.
3. Use the Prev and Next buttons to move back or forward within the trace to the events that matched your Scan Modes criteria.

   Or

   If you know the WaveScan table index (row) number of the event you wish to find, enter it in Idx.

PASS/FAIL Testing

PASS/FAIL testing allows you to define a set of qualifying conditions that an acquisition may "PASS" or "FAIL" when tested against, then take actions depending on the result.

- Mask testing compares sampled values to a pre-defined area of the grid (the "mask") to see if they fall within or without.
- Parameter comparison compares a measurement result to a pre-defined value (Param compare) or to another measurement (Dual param compare).

You can preconfigure multiple queries(Q*n*) using either method, which are then enabled by selecting them on the Pass/Fail dialog and defining what results constitute a "PASS" on the Actions dialog.

Mask Testing

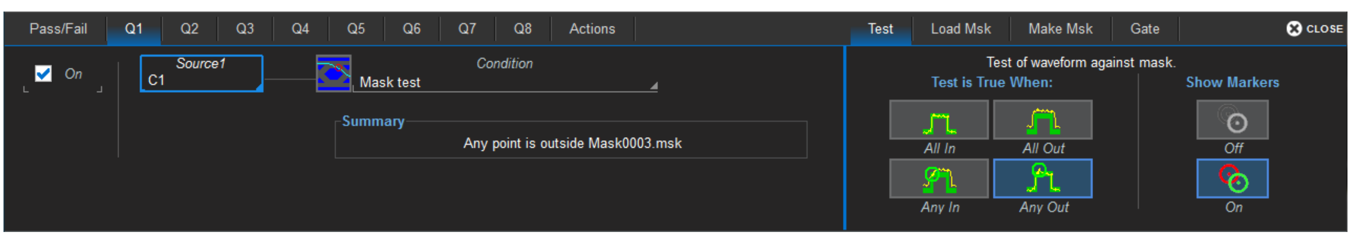
A mask defines an area of the grid against which a source Channel, Zoom, or Math trace is compared. Test conditions are associated with the mask, defining how the waveform is to be compared to the masked area (e.g., some/all values fall within, some/all values fall outside), and a pass or fail result is returned indicating the condition was found to be true or false.

Mask testing can be done using a pre-defined mask or a mask created from a waveform with user-defined vertical and horizontal tolerances. Some industry standard masks used for compliance testing are included with the oscilloscope software. The mask test can be confined to just a portion of the trace by the use of a measurement gate.

Access Mask Test Dialogs

1. Choose **Analysis > Pass/Fail Setup** to display the **Pass/Fail** dialog.
2. Touch the **Q*n* button** where you want to set up the query.
3. Select **Mask test**.

The Q*n* dialog opens with the Mask test condition selected. To the right are the **Test**, **Load Mask**, **Make Mask**, and **Gate** subdialogs where you manage all mask settings.



Make Mask

Use this procedure to create a new mask based on a live waveform. The mask covers the area of the waveform plus the boundaries you enter.

1. Open the **Make Mask** subdialog.
2. If desired, enter a new **Destination File Name** and path, or touch Browse and select a previous file to overwrite. The file name should end with the **.msk** extension.
3. Touch the **Ver Delta** and **Hor Delta** fields and enter boundary values using the pop-up Virtual Keypad or the front panel Adjust knob.
4. Touch **Make from Trace**.

Load Mask

Do this instead of Make Mask if you have a pre-defined mask file.

1. Open the **Load Mask** subdialog.
2. To use a saved .msk file, touch **File** and select the mask.
3. Check **View Mask** to display the mask over the trace.

Remove a Mask from the Display

Touch the **Delete All Masks** button on the Load Mask subdialog. Alternatively, open the **Q*n*** dialog where a particular mask is set, and on the **Load Mask** subdialog, touch **Delete**.

Set Gates

To limit the section of the waveform that is compared to the mask.

1. Open the **Gate** subdialog.
2. Enter the **Start** and **Stop** horizontal divisions that mark the section of the waveform to be tested. These can be a whole division or a fraction of a division. Divisions are numbered 1-*n* left to right.

> **Tip:** You can also drag the gate posts from the extreme left and right of the grid to the desired position.

Define "True"

1. Open the **Test** subdialog.
2. Select one of the conditions that, when true (yes), results in a pass or fail.
3. Optionally, turn **On** markers to show where on the waveform mask violations have occurred.

Param(eter) Compare

PASS/FAIL queries (Q*n*) can be configured to compare different parameter measurements (P*n*) to each other, or to a user-defined limit or statistical range. The Summary field shows the logic represented by the query. Check it to ensure your selections are producing the test you want.

Comparing a Measurement to a Limit

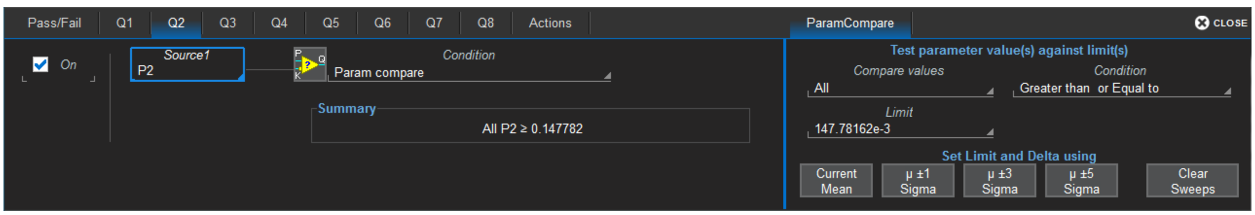
This method compares the result of a selected parameter to a user-defined limit or statistical range.

1. Choose **Analysis > Pass/Fail** to display the **Pass/Fail** dialog.

2. Touch the **Q*n* button** where you want to set up a query.

3. From the pop-up menu, choose **Param compare**.

4. On the Q*n* dialog, touch **Source1** and choose the parameter.



5. On the ParamCompare subdialog, touch **Compare Values** and select **All** (every value must be within limit to be "true") or **Any** (any value can be within limit to be "true").

6. Touch **Condition** and select a Boolean operator, then touch **Limit** and enter the value that completes the condition. The value entered acquires properties corresponding to the parameters being tested. For example, if you are testing a time parameter, the unit is seconds.

7. If you chose to set a Delta limit, also enter the **Absolute or % Delta** value. You may choose instead to set Limit and Delta using one of the buttons at the bottom of the dialog.

Comparing Parameter Measurements

This method compares one parameter to another, rather than a limit.

1. Choose **Analysis > Pass/Fail** to display the **Pass/Fail** dialog.

2. Touch the **Q*n* button** where you want to set up a query.

3. From the Pass/Fail Condition menu, choose **Dual Param Compare**.

4. On the left-hand Q*n* dialog, select the parameters to compare in **Source1** and **Source2**.

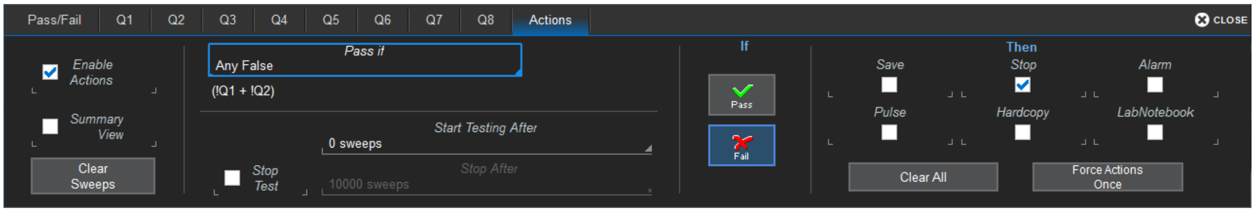
5. On the ParamCompare subdialog, choose to **Compare All** (every value must meet condition to be "true") or **Any** (any value can meet condition to be "true").

6. Touch **Condition** and select the Boolean operator that expresses the relationship between parameters (e.g., *All* P1 values must be *less than* P2).

Defining PASS/FAIL Tests

The queries define the conditions that, if met, would result in "true" but don't in themselves determine the test result. You must separately define what constitutes a "PASS" or a "FAIL" on the Actions dialog. Either result can be made to produce various additional actions, such as sending a pulse to another device.

1. After setting up the PASS/FAIL test qualifiers, open the **Actions tab**.



2. Optionally, check **Summary View** to see a running summary of results.
3. Select the test Pass criteria in **Pass If**.

Apply Actions

1. Check **Enable Actions** to turn on actions.
2. Under If, choose to apply actions if the result is a **Pass** or a **Fail**.
3. Under Then, choose all the actions to take:
   - Save **Waveform** file
   - **Stop** the test
   - Sound an **Alarm**
   - Emit a **Pulse** from the AUX OUT connector. When taking this action, also go to **Utilities > Utilities Setup > Aux Output** and choose to **Use Auxilliary Output For Pass/Fail**.
   - Capture the screen and process it according to your **Hardcopy** (Print) settings.
   - Create a **LabNotebook** Entry

Use the **Clear All** button to clear all the action checkboxes, or **Force Actions Once** to take action once regardless of the test results.

Delay Test

You can delay the start of a test by entering the number of sweeps to wait in **Start Testing After**.

Touch **Clear Sweeps** at any time to reset the test counter.

Stop Test

To stop the test following a specified number of sweeps, rather than a PASS or FAIL result, check the **Stop Test** box, then enter the desired number of sweeps in **After**.

Spectrum Analyzer

The SPECTRUM-1 and SPECTRUM-PRO-2R software options simplify the debugging of frequency-related effects, presenting a tool kit that mimics the interface of a traditional RF spectrum analyzer. You can start using the Fast Fourier Transform (FFT) with little or no concern about setting up an FFT. All spectral views can be displayed alongside other types of waveform traces for a comprehensive analysis of a signal's time and frequency response.

SPECTRUM-1 supports the creation and display of a single spectrum trace (SpecAn1) plus spectrogram (Spectro1). Optional peak labels and other markers add detail to the spectrum trace. The colored 2D or 3D spectrogram helps to visualize the frequency pattern.

In addition to all the features included with SPECTRUM-1, SPECTRUM-PRO-2R allows you to:

- Simultaneously generate two spectrums, both of which can be viewed as various types of spectral traces (Magnitude, Power, etc.) or as spectrograms
- Remove background noise
- Save either spectrum to a third, Ref(erence) spectrum
- Use Spectral mask testing to confirm that the frequency response of the DUT is within design limits

Choose **Analysis > Spectrum Analyzer** to access the software and begin to set up individual spectrums.

SPECTRUM-PRO-2R users can use the quick setup features once spectrum configurations are in place.

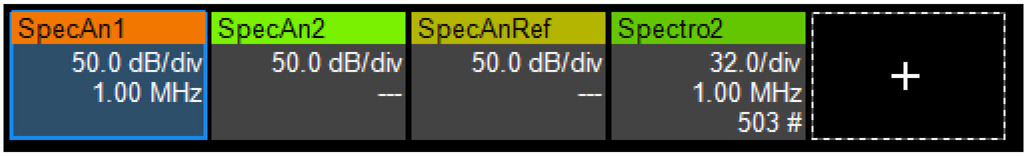
Frequency and Sample Rate

Spectrum Analyzer can only calculate frequency to half the acquisition sample rate. Therefore, the acquisition used as the spectrum source must be within the following limits.

Sample Rate	Lowest Frequency	Minimum Acquisition Length
10 GS/s and up		20 ns
5 GS/s	50 kHz	20 us
2.5 GS/s	25 kHz	40 us
2.5 MS/s	25 Hz	40 ms
500 kS/s	5 Hz	200 ms

Integration with Other Applications

Spectrums are now a type of trace that can be added to any other waveform display. If you are working in another oscilloscope application and turn on the Spectrum Analyzer, you will see the SpecAn dialogs at the bottom of the screen, but the rest of your waveform display will not change until you enable a spectrum or spectrogram. Then, a new SpecAn*N* or Spectro*N* descriptor box and trace will appear on the existing display.



Drag the new descriptor to whichever grid you'd like to move the spectrum/spectrogram and continue to modify it as you wish.

You will have to manually turn off any other traces you do not wish to view.

Quick Setup

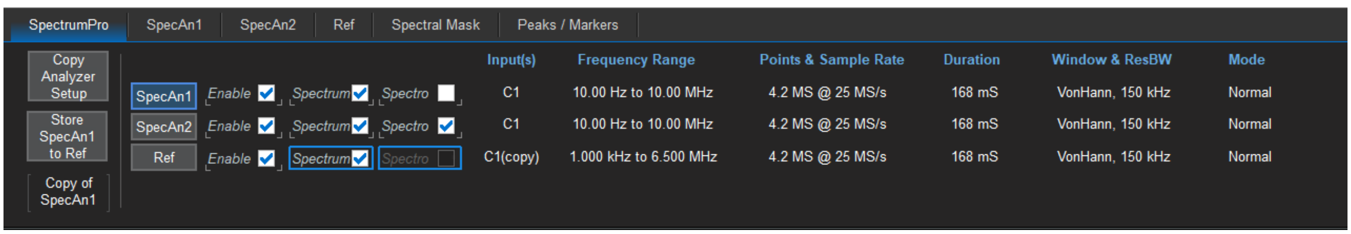
All features on the SpectrumPro dialog are only available with SPECTRUM-PRO-2R.

If you have previously configured spectrum traces, they can be quickly turned on/off or copied from the SpectrumPro dialog.

Turn On/Off Spectrum Traces

To turn on/off spectral views, simply check **Enable** and **Spectrum** or **Sectrogram**.

The settings used to calculate the view are shown next to the selections.



Copy Setup

A quick way to set up a new spectrum is to copy the setup of another spectrum. Select the **Copy Analyzer Setup** button, choose the **Copy From Lane** and **Copy To Lane**, then **Copy Now**.

Open the SpecAn*N* tab to modify any settings.

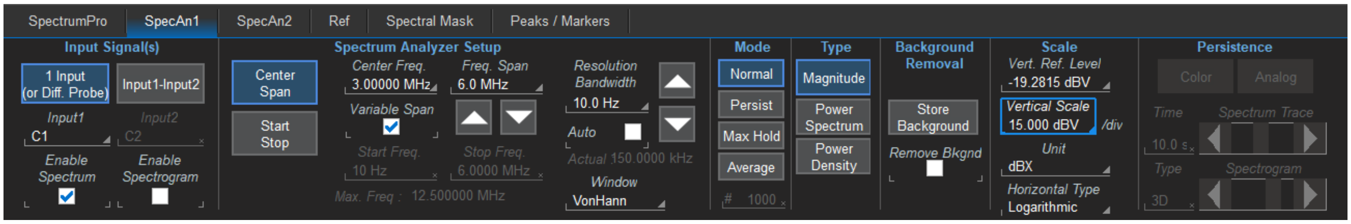
Store to Reference

Either spectrum can be stored as a Ref(erence) for future viewing and comparison with the other spectrums. Select the **SpecAn1** or **SpecAn2** button, then press **Store SpecAn*N* to Ref**.

Open the **Ref** dialog to modify the type and scale of the Reference lane display. Those settings that can be modified function exactly as do the equivalent SpecAn*N* dialog settings.

Set Up Spectrum

For the most basic spectrum set up, you need only choose the input signal and a frequency span. The software calculates the FFT and translates scope sample rate, memory, and acquisition length settings into frequency domain relevant units. The resolution bandwidth is automatically set to optimum resolution based on the acquisition record length.



Input Signal

Begin by selecting the signal(s) to be transformed.

To transform a differential probe signal or other single-ended input, touch **Input1** and choose the source.

To transform the difference between two single-ended inputs, select **Input 1-Input 2**, then choose the source for each input. This option eliminates the need to first set up a math trace to calculate the difference.

Note: The sources can be memory or math traces, as well as live channels, but it is not required to first create a math function for a differential signal.

Enable Spectrum displays an FFT of the last oscilloscope acquisition. This will be generated continuously if the oscilloscope is triggered in Normal or Auto modes.

Enable Spectrogram generates a 2D or 3D rendering of the historical data, up-to-256 spectra displayed in a vertically stacked display. See Spectrogram Display for instructions on controlling the spectrogram view.

Frequency Span

Note: The software always sets the oscilloscope sample rate equal to or higher than twice the frequency span entered.

Specify the frequency span to calculate by doing either of the following.

Select **Center Span**, then enter the **Center Freq.**(uency). A total **Freq.**(uency) **Span** will be calculated so that an equal span of frequencies appear on either side of the Center Freq. (i.e., a Center Frequency of 4 MHz will result in a total Frequency Span of 8 MHz). If you are using Linear scale, the Center Freq. will be at the center axis of the grid. (This will not necessarily be the case in Log scale).

You can adjust the total Freq. Span by using the Up or Down arrows below the field. Check **Variable Span** to make single increment adjustments with each touch; uncheck it to make stepped adjustments.

Alternatively, you can select **Start Stop**, then enter the absolute **Start Freq.** and **Stop Freq.** that constitutes the entire span.

Note: The Start frequency cannot be less than the Resolution Bandwidth setting.

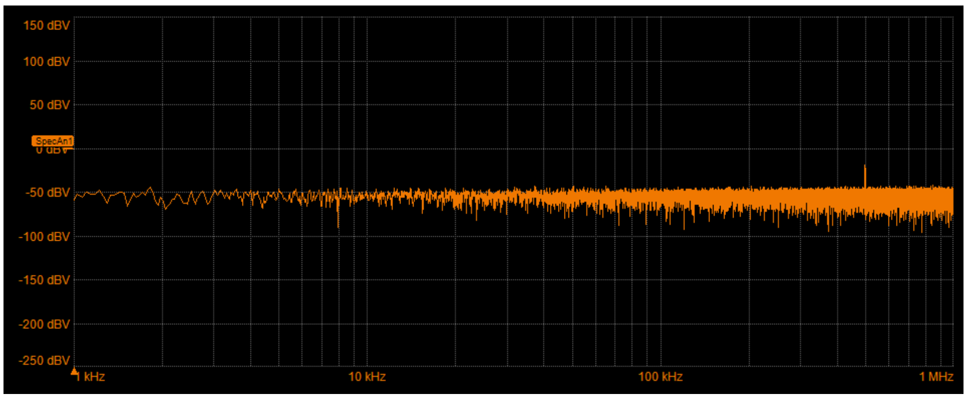
Resolution Bandwidth

Resolution Bandwidth is the frequency resolution, the spacing of samples in the spectrum trace. It is inversely proportional to the input record length.

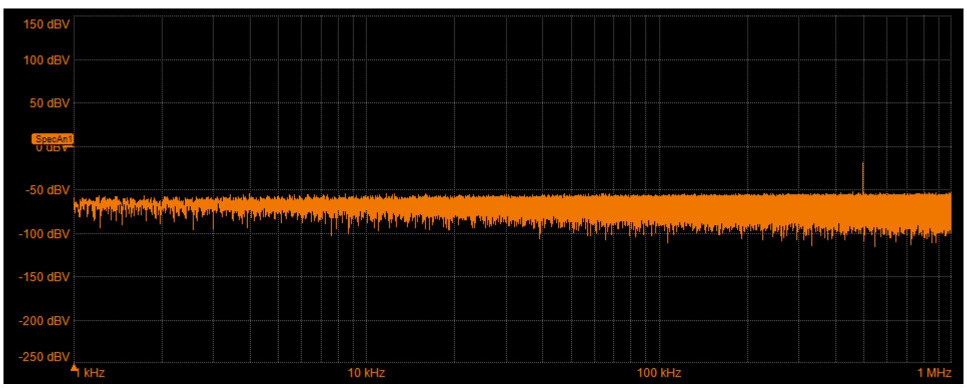
Changing Resolution Bandwidth is equivalent to changing the Timebase to increase or decrease memory. Reducing the resolution bandwidth will increase the available memory, enabling a faster update rate, but will decrease the resolution of the resulting spectrum trace.

By default, the spectrum is set to **Auto** Resolution Bandwidth, which will automatically set the highest resolution bandwidth possible based on the sampling rate of the input signal.

You can manually adjust this value by unchecking Auto and entering a new **Resolution Bandwidth** value.



Signal at 100 Hz Resolution Bandwidth.



Same signal and frequency span at 10 Hz Resolution Bandwidth.

FFT Window

If you think of an FFT as synthesizing a bank of parallel band-pass filters, weighting functions control the filter response shape and affect noise bandwidth as well as side lobe levels. Ideally, the main lobe should be as narrow and flat as possible to effectively discriminate all spectral components, while all side lobes should be infinitely attenuated. The window type defines the bandwidth and shape of the equivalent filter to be used in the FFT processing.

The choice of a spectral window is dictated by the signal's characteristics. Rectangular windows provide the highest frequency resolution and are useful for estimating the type of harmonics present in the signal. Because the rectangular window decays as a (SinX)/X function in the spectral domain, slight attenuation will be induced. Functions with less attenuation (Flat Top and Blackman-Harris) provide maximum amplitude at the expense of frequency resolution, whereas Hamming and Von Hann are good for general purpose use with continuous waveforms.

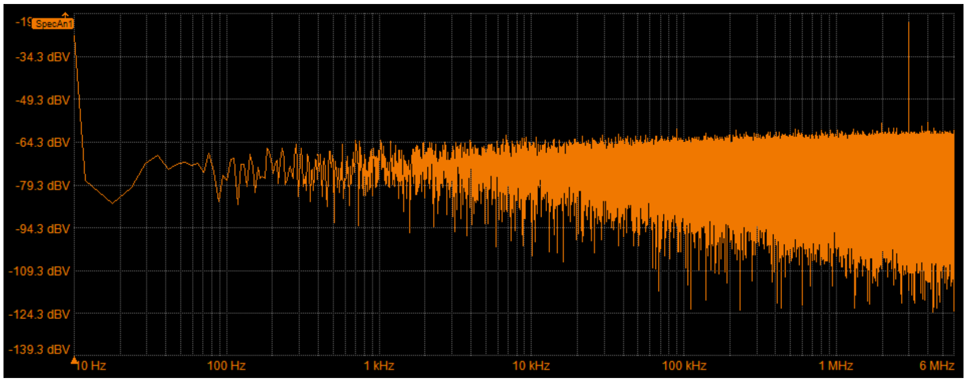
Window Type	Applications and Limitations
Rectangular	Normally used when the signal is transient (completely contained in the time-domain window) or known to have a fundamental frequency component that is an integer multiple of the fundamental frequency of the window. Signals other than these types will show varying amounts of spectral leakage and scallop loss, which can be corrected by selecting another type of window.
Hanning (Von Hann) & Hamming	Reduces leakage and improves amplitude accuracy. However, frequency resolution is also reduced.
Flat Top	Provides excellent amplitude accuracy with moderate reduction of leakage, but with reduced frequency resolution.
Blackman-Harris	Reduces leakage to a minimum, but with reduced frequency resolution.

FFT Window Filter Parameters				
Window Type	**Highest Side Lobe (dB)**	**Scallop Loss (dB)**	**ENBW (bins)**	**Coherent Gain (dB)**
Rectangular	-13	3.92	1.0	0.0
Von Hann	-32	1.42	1.5	-6.02
Hamming	-43	1.78	1.37	-5.35
Flat Top	-44	0.01	3.43	-11.05
Blackman-Harris	-67	1.13	1.71	-7.53

Operating Mode

Choose an operating **Mode** to determine how the spectrum trace is calculated and displayed.

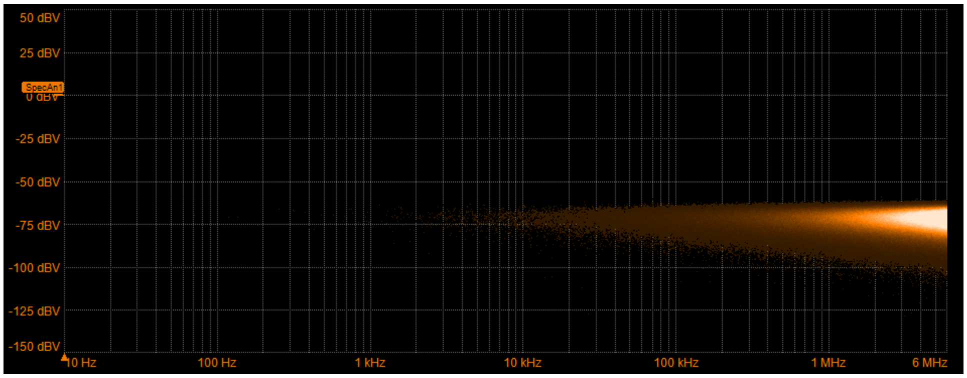
Normal (default) displays the Fast Fourier Transform of the input source. With a live oscilloscope input, this is a continuous transformation of time domain acquisitions.



Normal mode magnitude spectrum in log scale.

Persistence mode displays a history of spectra that are retained on screen, similar to the display of an RF spectrum analyzer. Use the Persistence controls at the right side of the dialog to select the persistence display type (Analog or Color) and adjust the level of saturation. See Persistence Display.

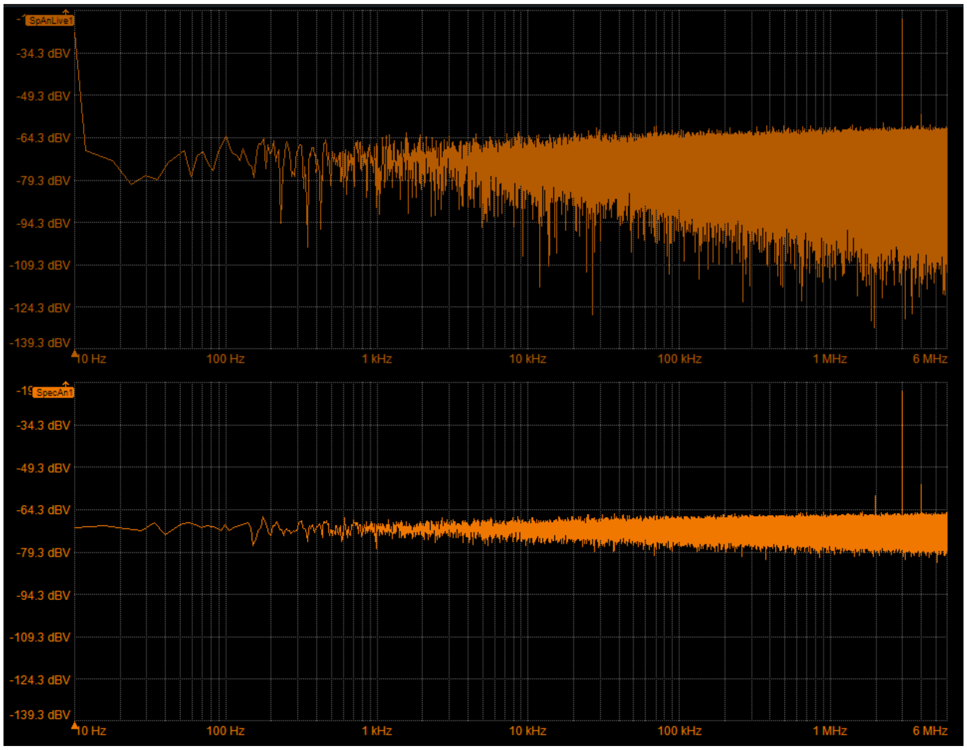
Note: Persistence mode requires a repetitive trigger to build the persistence map upon which the display is based.



Same signal as an analog persistence mode spectrum.

Max Hold displays a history of peak values across the frequency axis, essentially a "ceiling" calculator, useful for swept frequency measurements. As with persistence mode, the spectrum builds over multiple acquisitions. When this mode is selected, the normal FFT trace appears on screen as SpAnLive*N*, while the Max Hold trace appears as SpecAn*N*. Because the each new peak is held for the maximum amount of time possible, changes from the live trace only become apparent after long periods of time. The best use case for this operating mode is to leave the acquisition running for several hours or days to capture peak events.

Average displays a rolling average of the last **#** of FFT calculations. This mode effectively reduces the noise of the signal and displays more of the harmonic carrier detail. As with Max Hold mode, the normal FFT appears on screen as SpAnLive*N*, while the Average trace is presented as SpecAn*N*.



Live magnitude spectrum and Average spectrum of same input.

Spectrum Type

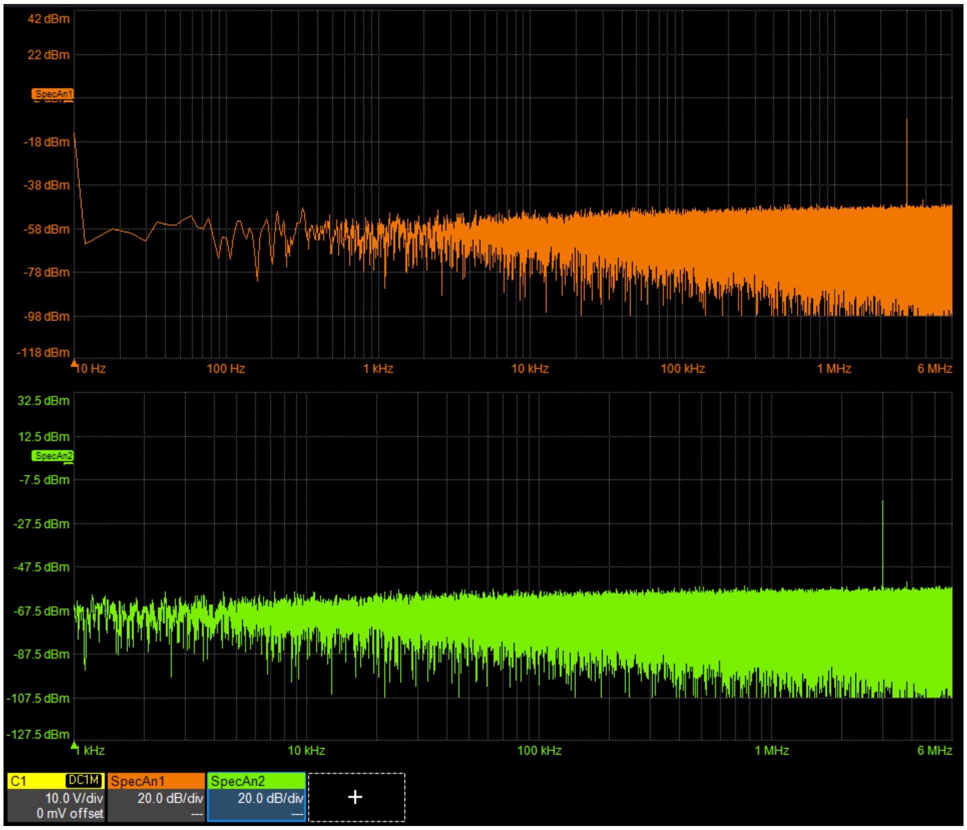
Choose the spectrum type, which will determine the default vertical unit of the trace.

Magnitude is the amplitude output in linear units (see examples above). This reads the peak value of a sine at each frequency.This trace is useful as an input to math operations, like adding and subtracting spectra. It may yield an unremarkable spectrum if viewed in linear units rather than decibels.

Note: A Magnitude spectrum will normally display the input source unit on the vertical axis. See Scale below for exceptions to this.

Power Spectrum is the power as a function of frequency using a logarithmic amplitude scale in units of decibels relative to 1 mW (dBm). This is a good general purpose view offering wide dynamic range so that you can see small spectral details.

Power Density is the power per unit frequency (power spectrum divided by the equivalent noise bandwidth of the FFT); the vertical scaling is also logarithmic with units of dBm. This view provides spectrum amplitude measurements that do not vary with the resolution bandwidth of the FFT, as they are normalized to the bandwidth.



Normal mode power (top) and power density (bottom) spectrums of same signal.
Note the change in frequency span and vertical reference, although the vertical scale/div is the same.

Background Removal

Background removal is available only with SPECTRUM-PRO-2R.

Background removal assists with removing noise due to a known interference signal, ADC spike or other unwanted effect that occurs in the signal path.

With the oscilloscope connected to DUT, but prior to starting the test signal, select **Store Background**. This will save a record of perturbations found in the channel. The example below shows the spikes found in a noisy channel, marked as Peaks 1 and 2.

Spectrum of background noise prior to background removal.

When the spectrum of the signal to be measured is displayed, check **Remove Bkgnd**. The Spectrum Analyzer will adjust the FFT calculation to account for the background noise.

Spectrum of input signal prior to background removal. A third peak belonging to the signal is identified.

Same spectrum after background removal. Now that the noise is suppressed, the peak attributed to the signal is ranked 1.

Scale

Vertical Ref. Level sets the amplitude represented by the top of the vertical axis, although the exact unit depends on the spectrum trace type and the input scale.

Vertical Scale sets the amplitude/div of the vertical axis.

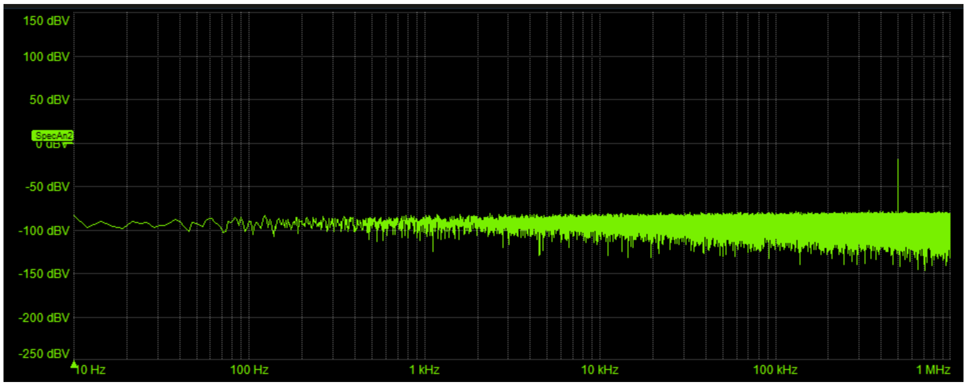
Tip: If you wish to change the unit of the vertical axis, do so before changing the reference level or scale. These values will revert to the default if the unit is changed to ensure that enough of a span is displayed that the spectrum appears on the grid.

Unit sets the units shown on the vertical axis.

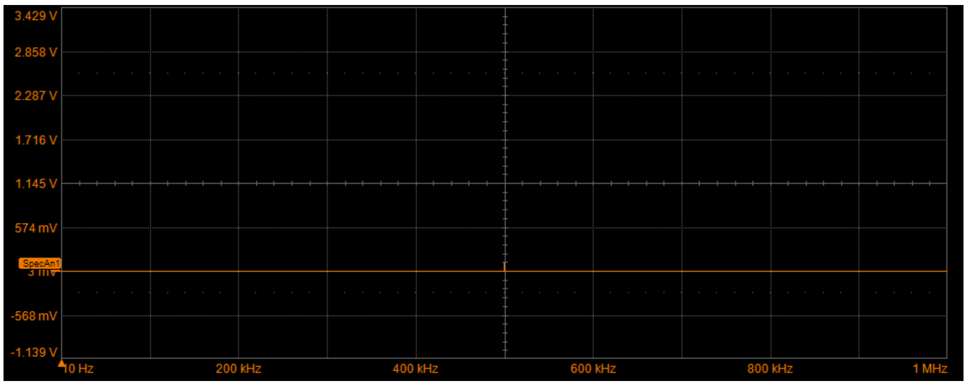
- Linear utilizes the same unit as is configured for the input signal. If you change the unit on the channel setup dialog (e.g., C5) that change is carried into the Spectrum Analyzer.
- Other selections convert the vertical unit to decibels at the same scale (dBX), milli-decibels (dBmX) or micro-decibels (dBuX).

Note: If the source is in units other than V, the vertical axis may not display milli- or micro-dB units (e.g., dBmT); the dBmX and dBuX selections will be disabled, in this case. Also, if the input unit is A, the vertical axis will simply say dB. However, the measured values will be rescaled according to your settings, and these will be displayed on the Spectrum Analyzer dialogs and tables.

Horizontal Type sets the horizontal axis to either Linear or Logarithmic scale.



Magnitude spectrum with dB units and Logarithmic horizontal type.



Magnitude spectrum of same signal and span in Linear units and Linear horizontal type.

Persistence Display

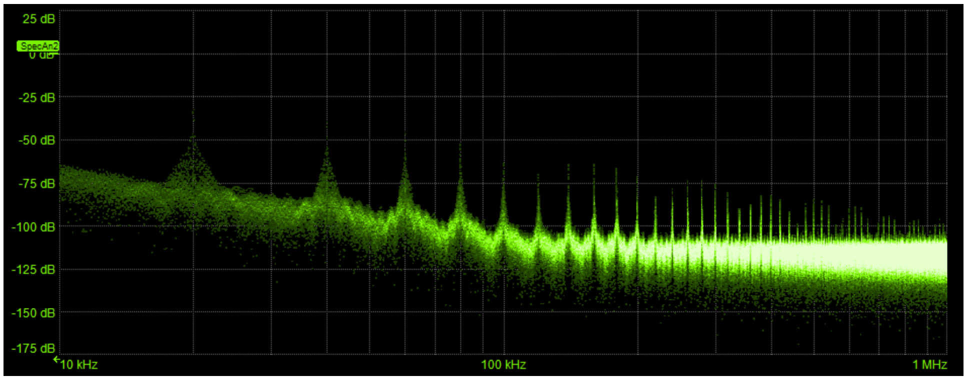
Persistence can be added to the spectrum or spectrogram views. Persistence shows the most frequent signal path as different intensities of the same color, or a graded spectrum of colors.

The Persistence feature retains spectra on the display for a set amount of time before allowing them to gradually "decay," similar to the analog-style display of old, phosphor screen oscilloscopes. The display is generated by repeated sampling of events over time and the accumulation of the sampled data into "persistence maps". Statistical integrity is preserved because the duration (decay) is proportional to the persistence population for each amplitude or time combination in the data.

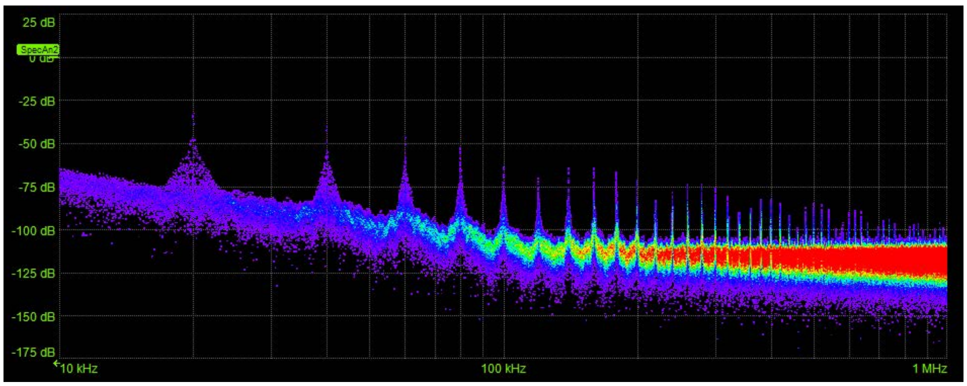
With **Analog** persistence, as a persistence data map develops, different intensities of the same color are assigned to the range between a minimum and a maximum population. The maximum population automatically gets the highest intensity, the minimum population gets the lowest intensity, and intermediate populations get intensities in between these extremes.

Color persistence works on the same principle as Analog persistence, but instead of different intensities (light vs. dark), variations in hue correspond to the occurrence of a frequency in the spectrum. "Hotter" hues indicate more frequent events, with red for maximum population and violet for minimum population. Color persistence is helpful for comparing amplitudes by seeking like colors among the traces, especially when used with the spectrogram view.

The saturation level, controlled by the **Spectrum Trace** or **Spectrogram** slider bar, represents a certain percentage of the total population. All populations above the saturation level are assigned the highest color intensity. At the same time, all populations below the saturation level are assigned the remaining intensities. Data populations are dynamically updated as data from new acquisitions is accumulated. A saturation level of 100% spreads the intensity variation across the entire distribution; at lower saturation levels the intensity will saturate (become brighter) at a lower population, making visible those events rarely seen at higher saturation levels.



Spectrum with analog persistence



Spectrum with color persistence

Spectrogram Display

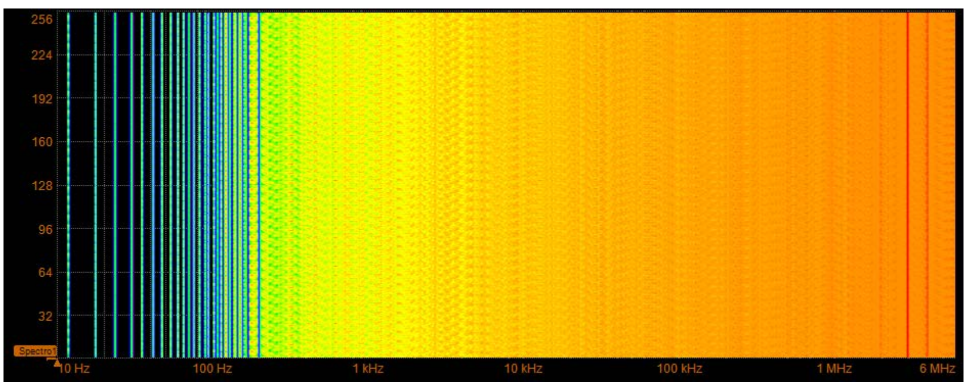
The spectrogram is a 2D or 3D rendering of the historical frequency data, up to 256 spectra in a vertically stacked display. Because spectrograms are type of persistence display, they require a repetitive trigger to build the persistence map. They can also be colorized like persistence spectrums to show different populations in different intensities or colors.

Spectrograms stop building after 256 acquisitions. Thereafter, they are calculated on the last 256 acquisitions in the buffer.

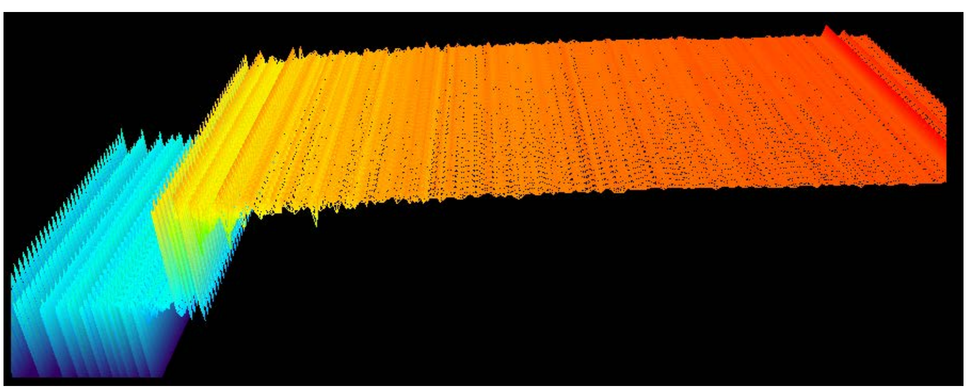
1. Set up the spectrum trace as usual.
2. On the SpectrumPro or SpecAn*N* dialog, select the **Enable Spectrogram** checkbox.

   You may need to retrigger the acquisition to begin building the persistence map.
3. Choose a spectrogram **Type** of 2D or 3D.

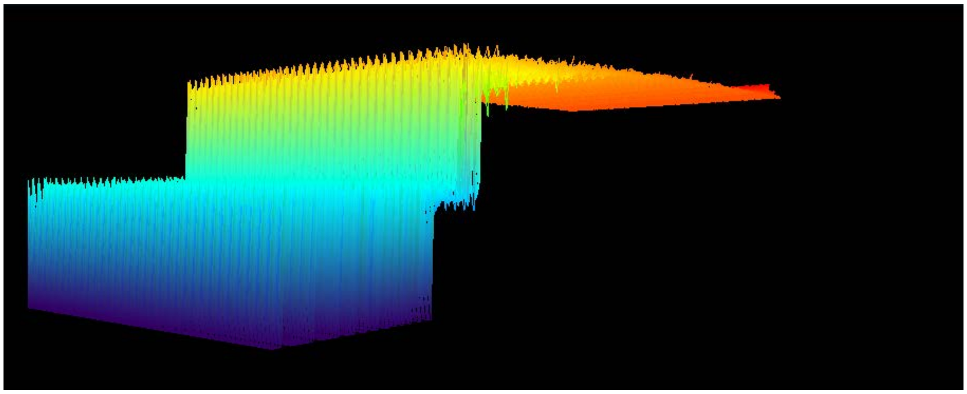
   The 3D spectrogram view can be rotated on any of its axes by grabbing an edge of the display with a finger or pointer and dragging. The display is active when you see the cross-hatch symbol.
4. Choose how to colorize the spectrogram by selecting the **Analog** or **Color** Persistence button.
5. Move the **Spectrogram** slider to increase/decrease persistence saturation level.



2D color spectrogram



Same signal as 3D color spectrogram



Same 3D spectrogram, rotated.

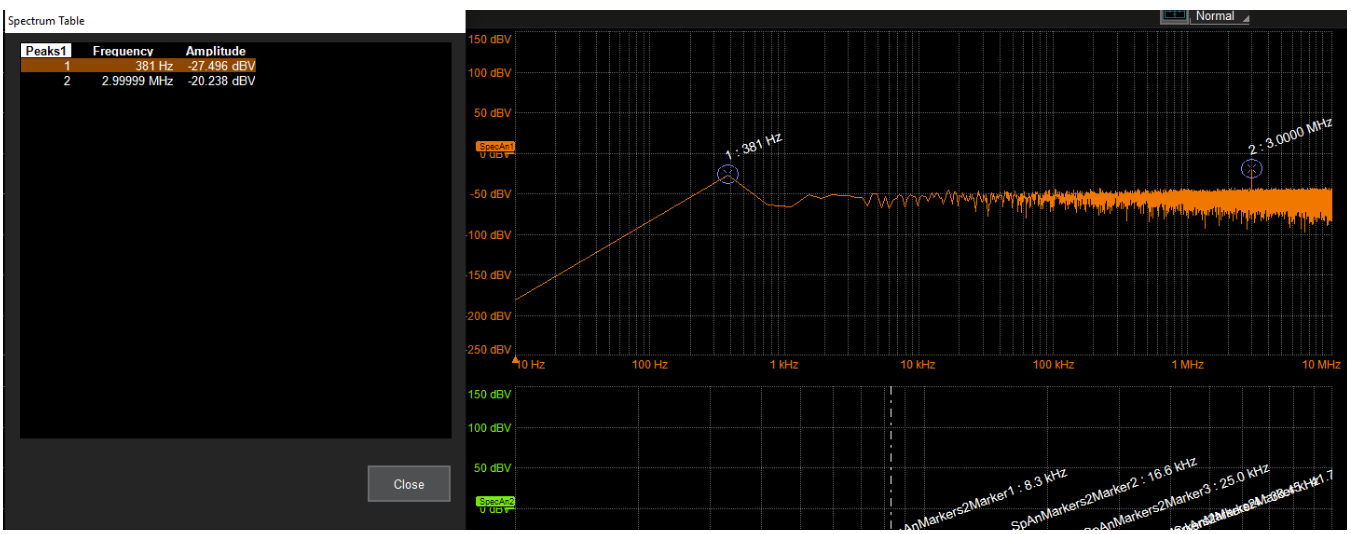
Peaks and Markers

Two special annotations help you find frequencies or amplitudes of interest in the spectrum trace: Peak frequencies and user-defined Markers.

Begin by opening the Peaks/Markers tab and selecting the button for the spectrum (SpecAn*N* or Ref) to annotate from the left side of the dialog.

Peaks

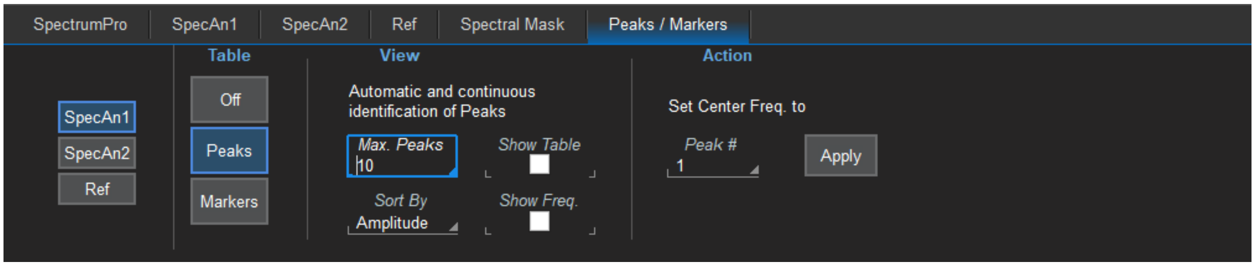
The peaks features is an automatic and continuous identification of waveform peaks. As the range of frequencies measured changes, peak values are recalculated and markers are moved. The values tabulated are always absolute values for the marked peak.



Peaks identified on spectrum.

To show peaks:

1. With the correct **SpecAn*N*** selected and visible, select the **Peaks** button.



2. Enter the **Max.**(number of) **Peaks** to mark. Circular markers appear over the trace, representing the x peak measurements.

3. Choose to **Sort By** amplitude or frequency. Peaks will be renumbered from highest (1) to lowest (x) corresponding value.

4. To display the frequency of the peak on the marker, check **Show Frequency**.

5. For a tabular listing of peaks, check **Show Table**.The frequency and amplitude of each peak will appear in a pop-up dialog over the grid display.

> Note: Only those peaks visible on the display will be listed in the table, even if other peaks have been identified in the acquisition.

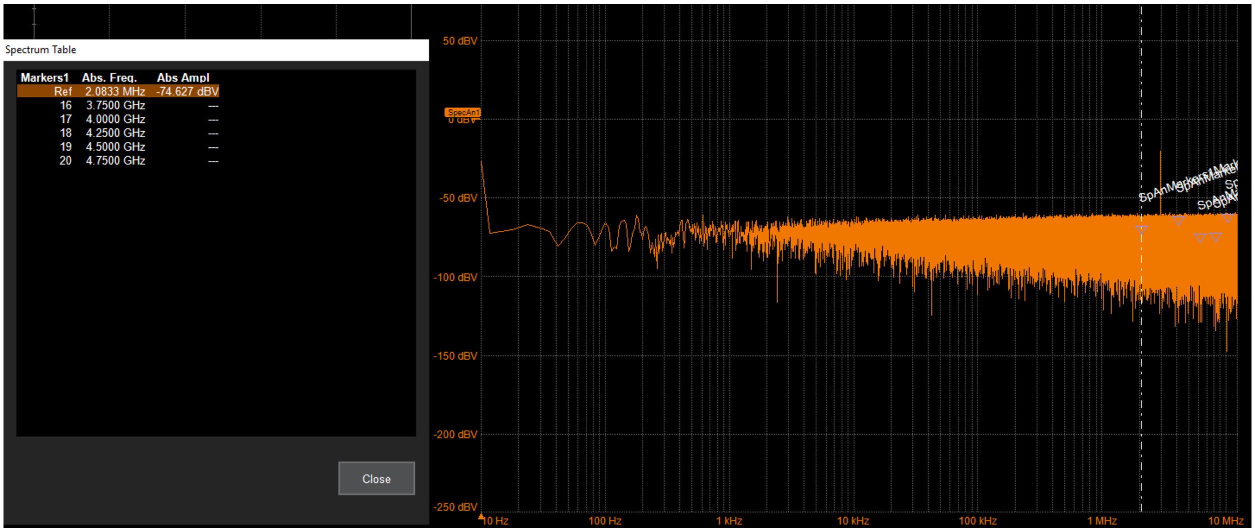
6. Optionally, reposition the display by choosing a peak to **Set Center Freq. to**. Enter the **Peak #** and touch **Apply**. If using Linear horizontal scale, the spectrum will shift to place that peak at the center of the grid.

Markers

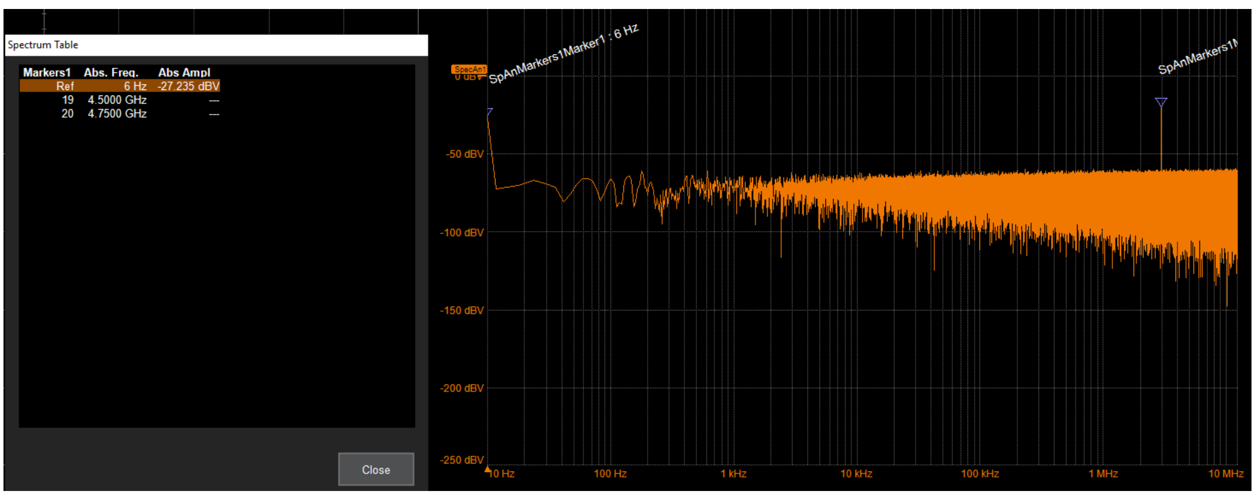
You can apply up-to-20 markers to frequencies of interest, which are indicated by blue triangles over the trace. Marker 1 is always the Ref(erence) Marker, indicated by a thick, white cursor line. Delta measurements of the other markers are made in reference to this frequency.

Three pre-defined sets of markers can be quickly displayed over the spectrum to save the work of manually placing each marker.

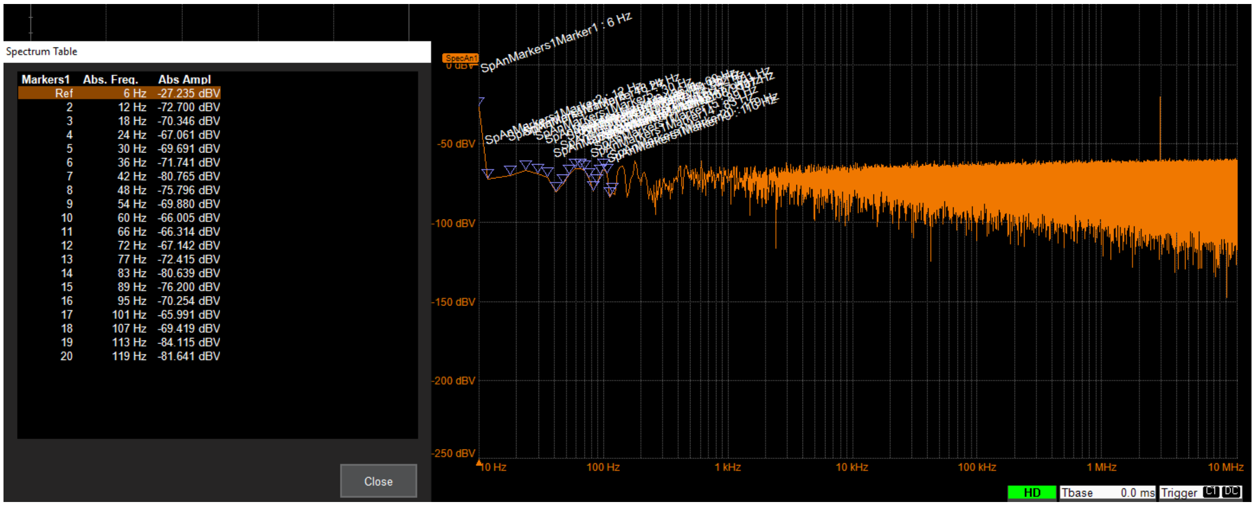
- Set 5 Default Markers marks five frequencies spread evenly over the spectrum. This is a good way to begin placing markers, moving them to the desired points on the spectrum using any of the marker controls (below).



- Set Markers on Peaks marks the peak amplitudes in the spectrum, same as Peaks, only the annotation is different and will not automatically update with each new acquisition, only when you move the marker.



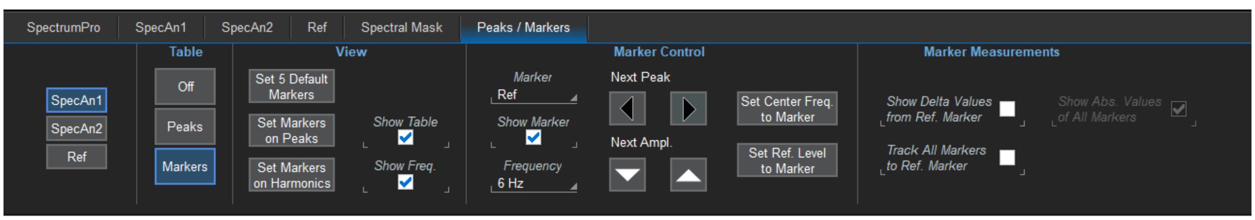
- **Set Markers on Harmonics** marks the fundamental frequency and its harmonic content.



If you turn off markers, the display will resume the set you last selected when they are turned on again.

To apply markers:

1. With the correct **SpecAn*N*** selected and visible, select the **Markers** button.



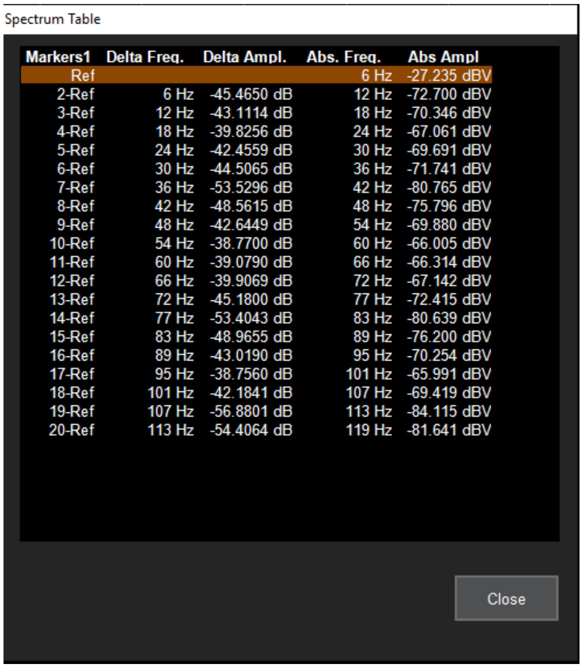
2. Choose a set of markers to view by clicking the button, or manually select the number of the **Marker** to place and check **Show Marker**. Use the marker controls (below) to move the marker to the desired position.

> Note: The marker will be placed on the last sample closest to the frequency entered or calculated from the overall span. For example, a marker set on Frequency 3 MHz may be placed at 2.99 MHz. This will be reflected in the frequency shown on the marker and in the marker readout table.

3. To display the frequency represented by the marker on the waveform, check **Show Frequency**.

4. For a tabular listing of markers, check **Show Table**.The amplitude and frequency of each marker location will appear in a pop-up dialog over the grid display. You can optionally add the delta from

the Ref marker frequency to the table. See Marker Measurements (below).



Spectrum Table

Markers1	Delta Freq.	Delta Ampl.	Abs. Freq.	Abs Ampl
Ref			6 Hz	-27.235 dBV
2-Ref	6 Hz	-45.4650 dB	12 Hz	-72.700 dBV
3-Ref	12 Hz	-43.1114 dB	18 Hz	-70.346 dBV
4-Ref	18 Hz	-39.8256 dB	24 Hz	-67.061 dBV
5-Ref	24 Hz	-42.4559 dB	30 Hz	-69.691 dBV
6-Ref	30 Hz	-44.5065 dB	36 Hz	-71.741 dBV
7-Ref	36 Hz	-53.5296 dB	42 Hz	-80.765 dBV
8-Ref	42 Hz	-48.5615 dB	48 Hz	-75.796 dBV
9-Ref	48 Hz	-42.6449 dB	54 Hz	-69.880 dBV
10-Ref	54 Hz	-38.7700 dB	60 Hz	-66.005 dBV
11-Ref	60 Hz	-39.0790 dB	66 Hz	-66.314 dBV
12-Ref	66 Hz	-39.9069 dB	72 Hz	-67.142 dBV
13-Ref	72 Hz	-45.1800 dB	77 Hz	-72.415 dBV
14-Ref	77 Hz	-53.4043 dB	83 Hz	-80.639 dBV
15-Ref	83 Hz	-48.9655 dB	89 Hz	-76.200 dBV
16-Ref	89 Hz	-43.0190 dB	95 Hz	-70.254 dBV
17-Ref	95 Hz	-38.7560 dB	101 Hz	-65.991 dBV
18-Ref	101 Hz	-42.1841 dB	107 Hz	-69.419 dBV
19-Ref	107 Hz	-56.8801 dB	113 Hz	-84.115 dBV
20-Ref	113 Hz	-54.4064 dB	119 Hz	-81.641 dBV

Close

Marker Control

Use the Marker Controls to place, move or hide a particular marker after it's initial placement using one of the "Set..." buttons. All values in the Marker table and on the Marker annotation will update dynamically as the marker is moved.

First, enter the **Marker** number. A white cursor line now bisects that marker, and all other marker controls apply to it.

> **Tip:** The easiest way to move the marker is to drag the cursor line to a new frequency. The tabular values are recalculated as the marker moves.

To show/hide the marker, select or deselect the **Show Marker** checkbox.

To move the marker to:

- The **Next Peak** in either direction, touch the **Left/Right Arrow**.
- The peak with the **Next Amplitude** (higher or lower), touch the **Up/Down Arrow**.
- A specific **Frequency**, enter the new value.

Set Center Freq. to Marker will make that marker value the center frequency of the span displayed.

To use a particular marker location as the new reference level, select **Set Ref. Level to Marker**. All delta measurements are now referenced to this frequency.

Marker Measurements

You can modify the measurements displayed with each marker annotation and in the marker table by selecting/deselecting the checkboxes at the right of the dialog:

- **Show Abs. Value of All Markers** shows the absolute frequency value at the location of each marker. This is the default and is automatically selected when you turn on markers. If you choose to show Delta, you can optionally turn off the display of absolute values to view only the delta values.
- **Show Delta Values from Ref. Marker** shows the difference in frequency value between this marker location and the reference marker.
- **Track All Markers to Ref. Marker** locks all markers at their current delta from the Ref Marker. When the Ref marker is dragged, all the markers are moved as a group. This is useful for finding interesting harmonics in the spectrum.

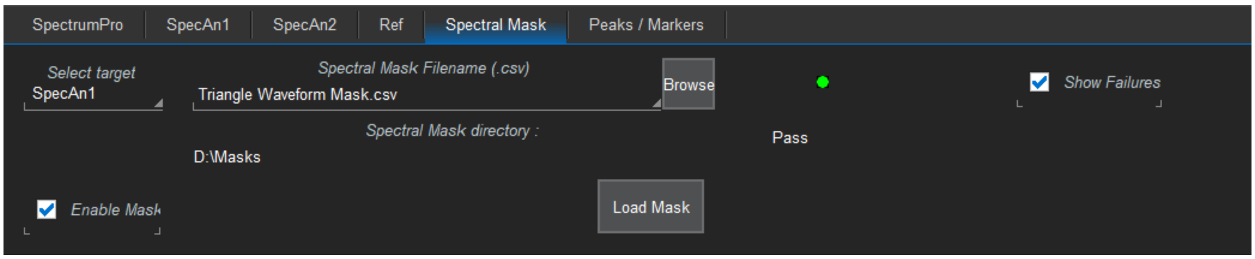
Spectral Mask Testing

Spectral mask testing is available only with SPECTRUM-PRO-2R.

Spectral mask testing enables you to test that the frequency response of the DUT is within design limits. You generate data masks in a simple, .csv format and upload them for use within Spectrum Analyzer. The software will automatically calculate if the spectrum "passes" within the limits defined by your mask, optionally marking any violations on the display.

Set Up Mask Test

1. Copy your mask file to oscilloscope D:\Masks (for ease of selection), or to a USB drive that can be connected to the oscilloscope.
2. Open the **Spectral Mask** dialog.



3. **Select Target** spectrum to test.
4. **Browse** to and select the spectral mask file to apply.
5. With the file loaded in the Spectral Mask Filename field, check **Enable Mask** to begin pass/fail testing.
6. Optionally, check **Show Failures** to mark violations on the spectrum trace. Circular red markers appear where the signal violates the mask.

You will see "Processing..." in the message bar while the mask test is calculating. Within a few seconds, a purple mask should appear over the spectrum trace marking the amplitudes and frequencies that are outside the mask limits. If you have chosen to show failures, any samples that occur in this masked area are marked by red circles.

Note: Masks and mask markers will appear in screen captures and reports saved with LabNotebook files, but they cannot be recalled ("Flashback") with LabNotebook waveform data, as they are calculated from a separate file.

Mask File Format

Masks are created as .csv files.

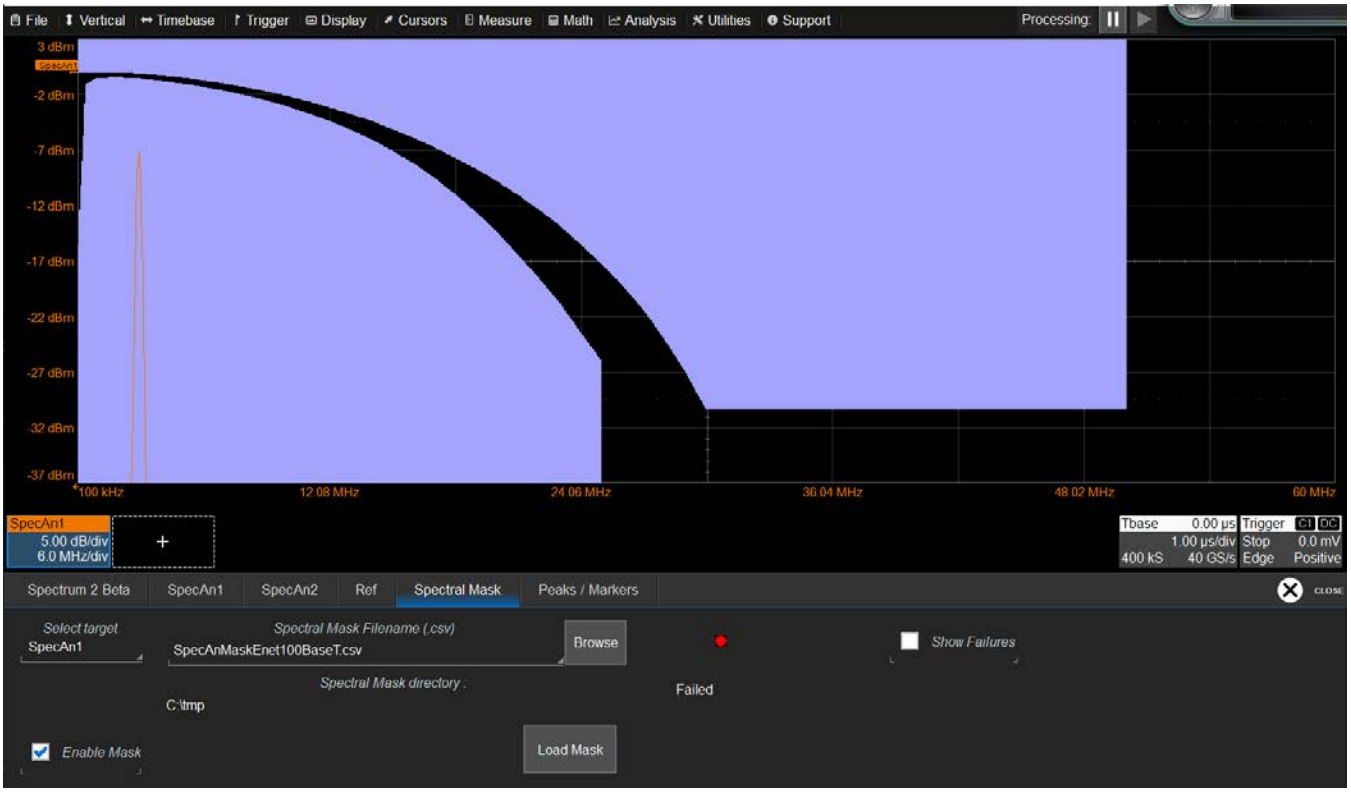
- Any lines that start with a quote are handled as comments (e.g., rows 1 and 2 below).
- The first line read (row 3 below) contains the X and Y axis units. PSD is currently ignored, do not add it.
- The second line read (row 4 below) contains the keyword titles Frequency, Upper and Lower.
- Each subsequent line contains the upper and lower passing limits for each frequency. The value of upper should always represent a position above lower on the graticule.

Because the mask identifies data values, rather than pixel locations, the mask will automatically adjust with changes to the waveform scale, including changes from linear to log scale.

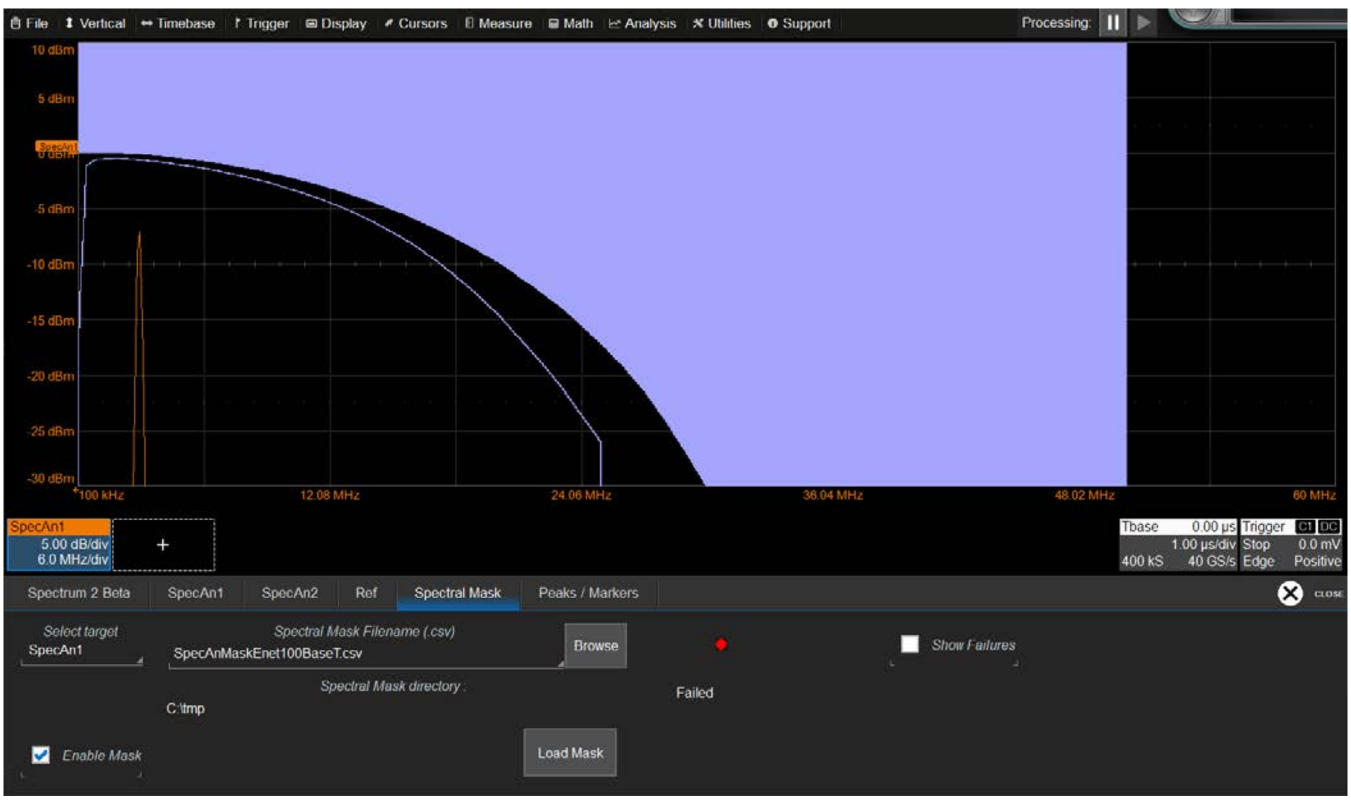
Note: The proper calculation of the mask area depends on the fill point being visible on screen. Rescaling, delay or offset that shifts this point off the grid will result in the area not filling, as in the second example below. Because the test is a mask test, if the mask fails, the Pass/Fail results will not be calculated properly. Be sure that your mask remains visible outside of your valid limits.

	A	B	C	D
1	"Line 1: Xin, [Hz|KHz|MHz|GHz],Yin,[V,A,T,H,dB,dBm,dBA,dBT etc], [PSD] - add PSD if vertical units are per Hz			
2	"Line 2: keyword titles. Must be: Frequency, Uppper,Lower . Lines that start with a quote are ignored.			
3	Xin	MHz	Yin	dBm
4	Frequency	Upper	Lower	
5	0.1	0	-17.88	
6	0.2	0	-13.49	
7	0.3	0	-9.09	
8	0.4	0	-4.7	
9	0.5	0	-1.13	
10	0.6	0	-1.01	
11	0.7	0	-0.9	
12	0.8	0	-0.78	
13	0.9	0	-0.66	
14	1	0	-0.59	
15	2	0	-0.52	

Sample spectral mask file



Spectral mask prior to rescaling Y axis. Note how the test fails because the peak falls within the mask.



Spectral mask after rescaling Y axis. The bottom of the mask has dropped out because the fill point is off screen.

Saving Data (File Functions)

The File menu functions allow you to save and recall setups, waveform data, table data, screen captures, LabNotebooks, and reports. You can use Print or E-mail to share these files.

LabNotebook is Teledyne LeCroy's proprietary tool for capturing a composite file containing waveform data, oscilloscope setups and display, which can be output to a preformatted report. An important feature of LabNotebook is Flashback, which enables you to restore an acquisition (and the setups used to create it) to the oscilloscope screen simply by recalling a LabNotebook entry.

Save

Oscilloscope setups (configurations), waveform data, tabular data, and the display can all be saved in multiple formats.To save them all as a composite LabNotebook file, see Save LabNotebook.

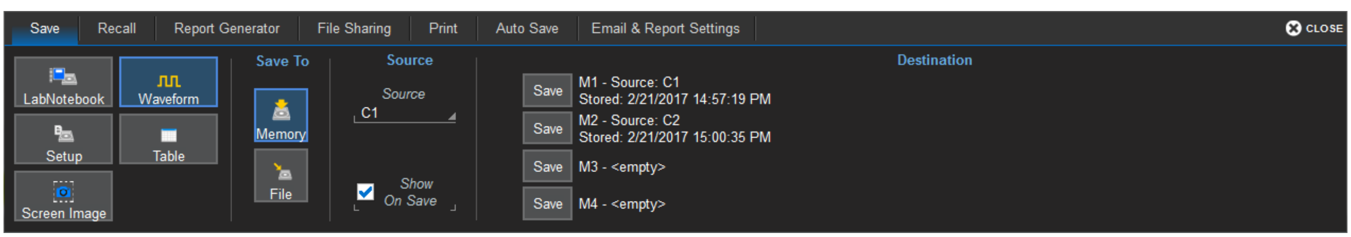
Note: We strongly recommend that you stop acquisition before saving waveforms or LabNotebooks, especially when running in Auto trigger mode. If you do not first stop, there is no guarantee that what you are capturing is what you want to save.

Save Waveform

The Save Waveform function saves trace data either to internal memory (M1-M*n*) or to a file in text or binary format. The source waveform can be any type of trace; a channel, math function, zoom, or even another memory. Waveform files can be recalled into an internal memory, from where they can be restored to the display.

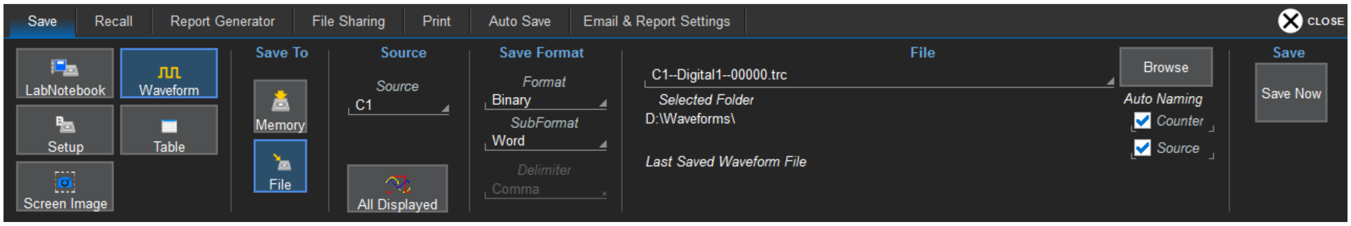
Note: Only traces saved in Teledyne LeCroy proprietary formats can be recalled to the display. Use the Binary format (.trc) for analog waveforms, or WaveML (.xml) for digital waveforms.

Save Waveform To Memory



1. From the menu bar, choose **File > Save Waveform**.
2. Choose to Save To **Memory**.
3. Choose the **Source** trace you are saving.
4. Optionally, select **Show On Save** to display the memory trace with its source as soon as it is saved.
5. Touch the **Save** button directly to the left of the selected **Destination** memory slot. The date/time stamp of the saved memory will appear at that location.

Save Waveform To File

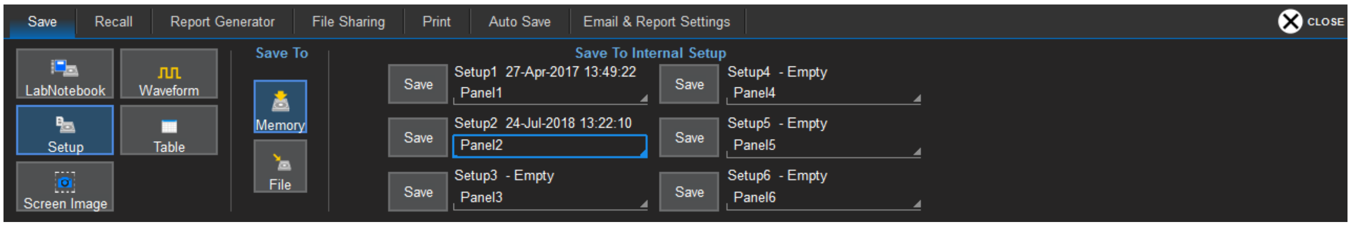


1. From the menu bar, choose **File > Save Waveform**.
2. Choose to Save To **File**.
3. Choose the **Source** waveform. To quickly save all displayed waveforms separate trace files, touch the **All Displayed** button.
4. Select a file **Format**. **Binary** is Teledyne LeCroy's .trc file format. Binary results in the smallest possible file size. Binary files can be converted to ASCII using Teledye LeCroy utilities such as WaveStudio. **WaveML** is a proprietary .xml format used to save persistence maps, eye diagrams, histograms, and digital traces. This option is only active if the source is compatible.
5. If your Format is ASCII, touch **Delimiter** and select a character from the pop-up menu.
6. Depending on your Format selection, you may also need to specify a **SubFormat:**
   - **Word** (Binary) represents samples using 16 bits. Always use this unless Byte mode is "pre."
   - **Byte** (Binary) represents samples using 8 bits. This option can result in a loss of resolution.
   - **Auto** (Binary) looks at the data and automatically selects either Word or Byte subformat.
   - **Amplitude only** (Text) includes amplitude data for each sample, but not time data.
   - **Time and Amplitude** (Text) includes both types of data for each sample.
   - **With Header** (Text) includes a file header with scaling information.
7. The system will auto name the file <Source>-Trace-<Counter>. To change the file name or Selected Folder, enter the full path and name in **File**, or use the File Browser. By default, trace files are saved to D:\Waveforms on the instrument hard drive.
8. If you do not want to use the **Source** prefix or **Counter** number, deselect them.
9. Touch **Save Now**.

Save Setup

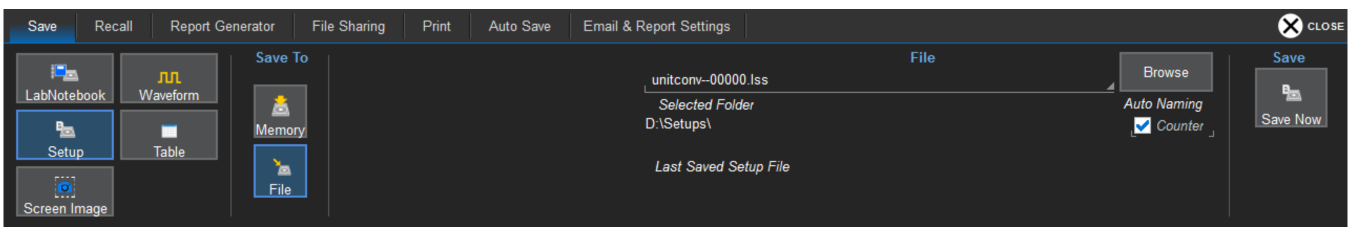
You can quickly save oscilloscope configurations to one of the internal setup panels or to a LeCroy System Setup (.lss) file, a text-based Automation program. Setups can be easily recalled to restore the oscilloscope to the saved state.

Save Setup to Memory



1. From the menu bar, choose **File > Save Setup**.
2. Choose to Save To **Memory**.
3. If desired, touch one of the **Setup** slots and enter a name for the memory. The default name will be Panel*n*. Try to select an empty slot, as anything currently in it will be overwritten.
4. Touch the **Save** button directly to the left of the selected Setup slot. The save date/time is displayed above the **Setup** field.

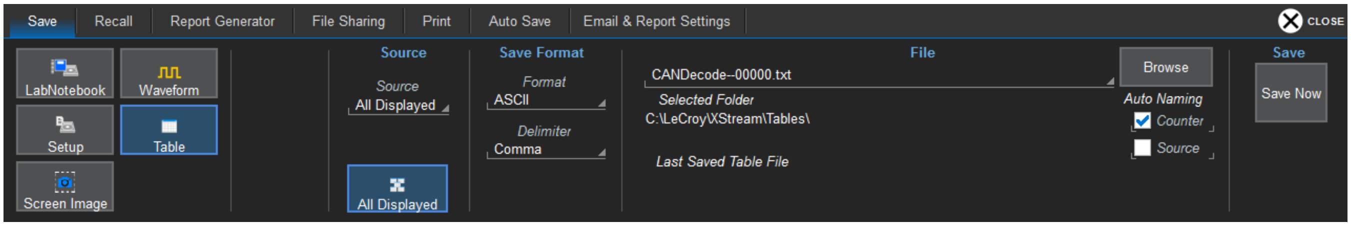
Save Setup to File



1. From the menu bar, choose **File > Save Setup**.
2. Choose to Save To **File**.
3. To change the file name or Selected Folder, enter the full path and name in **File**, or use the File Browser. By default, setup files are saved to D:\Setups on the instrument hard drive.
4. If you do not want to use the **Counter** number, deselect it.
5. Touch **Save Now**.

Save Table

The Save Table function saves tabular measurement data displayed on screen to an Excel or ASCII file.

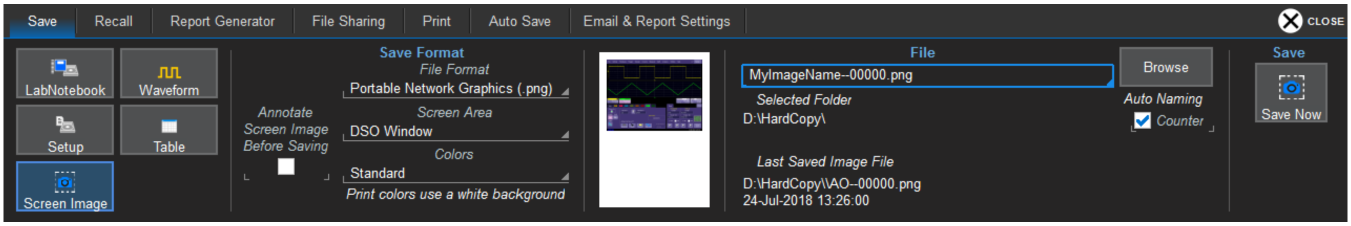


1. From the menu bar, choose **File > Save**, then choose **Table**.
2. To save only one of the tables displayed, touch **Source** and navigate to the selection.

   By default, data from all visible tables are saved to separate files. To quickly restore this selection, touch the **All Displayed** button.
3. Choose a format of **ASCII** (.txt) or **Excel** (.csv). If you selected **ASCII** format, also touch **Delimiter** and choose a character.
4. The system will auto name the file "<source>-Table-<counter>". To change the file name or Selected Folder, enter the full path and name in **File**, or use the File Browser. By default, table files are saved to D:\Tables on the instrument hard drive.
5. If you do not want to use the **Source** or **Counter** number, deselect them.
6. Touch **Save Now**.

Save Screen Image

The full touch screen display or selected portions of it can be captured, saved to an image file, and marked with custom annotations.

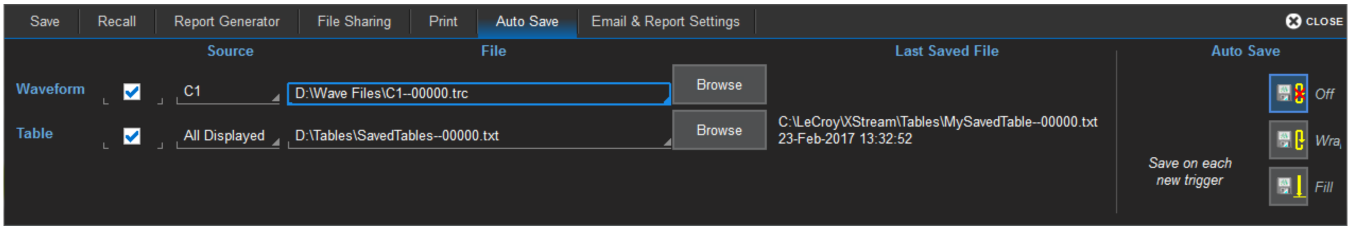


1. From the menu bar, choose **File > Save**, then choose **Screen Image**.
2. Choose a **File Format** of .JPEG, .PNG, .TIF, or .BMP.
3. To mark up the screen image using the drawing tools, select **Annotate Screen Image Before Saving**.
4. Select the **Screen Area** captured:
   - Grid Area Only includes any visible waveform grids, tables, and descriptor boxes.
   - DSO Window includes the above plus any open dialogs, menu bar, and message bar.
   - Full Screen is the full Windows display, including other visible applications and desktop.
5. Choose the **Colors** used in the capture:
   - Standard uses the screen colors on a black background as it appears on the instrument.
   - Print (default) uses your print color palette (set in Preferences) on a white background to save ink.
   - Black & White captures the image in grayscale.
6. To change the file name or Selected Folder, enter the full path and name in **File**, or use the File Browser. By default, image files are saved to D:\Hardcopy on the instrument hard drive.
7. Touch **Save Now**.
8. If you've turned on annotations, use the Drawing toolbar to mark up the screen image. Click **Done** when finished.

Auto Save

Data that appears on the oscilloscope display, such as waveforms, measurement readouts and decoder data, can be very dynamic and difficult to read from the oscilloscope unless you stop the acquisition.

Auto Save enables you to automatically store waveform and table data to file with each new trigger. The file can later be recalled to the oscilloscope or saved permanently to external storage.



1. Choose **File > Save** and open the **Auto Save** dialog.
2. Select to save **Waveforms** and/or **Tables**.
3. Select the **Source** from which to save. The All Displayed selection will create a separate file for each trace or table displayed.
4. By default, waveform files are stored in D:\Waveforms, and tables are stored in D:\Tables on the instrument hard drive. Optionally:
   - Enter a different root path and name in **File**.
   - Touch **Browse** to open the File Browser and add or remove the source prefix and counter suffix.
5. Choose one of the **Auto Save** options: **Wrap** (old files overwritten when buffer filled) or **Fill** (no files overwritten, excess is truncated).

   Tip: If you have frequent triggers, it is possible you will eventually run out of storage space. Choose Wrap only if you're not concerned about files persisting on the instrument. If you choose Fill, plan to periodically delete or move files off the instrument, or you will lose new data.

6. To enable these selections, choose **Configure Auto Save** from the pop-up dialog that appears.
7. As soon as there is a trigger, files will begin to be saved. Choose **Disable Auto Save** to stop saving files while the acquisition is in process.

   You can also choose **Off** on the Auto Save dialog to disable this feature in between acquisitions.

Recall

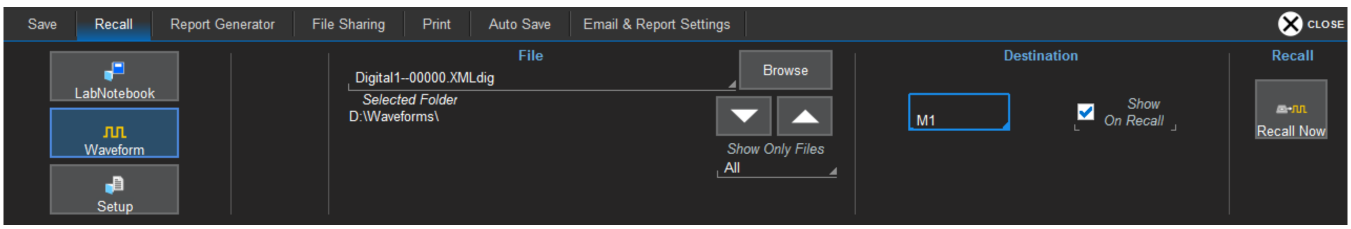
Setups saved to internal memory or to .LSS file can be recalled to restore the oscilloscope to the saved state.

Waveform data stored to .TRC or .XML files can be recalled into a memory and from there restored to the screen.

To recall and modify saved LabNotebook files, see Recall LabNotebook.

Recall Waveform

Note: Only files saved in Binary (.trc) or WaveML (.xml) format can be recalled to the touch screen.



1. Choose **File > Recall Waveform** from the menu bar.
2. Touch **Browse** and navigate to the file.

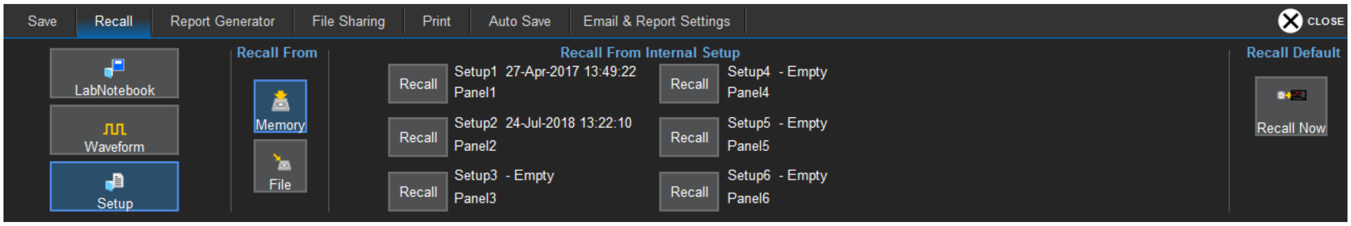
   If the file is in the Selected Folder, you can use the **Up /Down Arrows** to cycle through the available files until the desired file is selected.

   Optionally, touch **Show only files** to apply a search filter (channels, math functions, or memory) to the list of available files.
3. Select a **Destination** memory slot into which to recall the file.
4. Mark **Show on Recall** to display the trace on the grid.
5. Touch **Recall Now**.

Recall Setup

Choose **File > Recall Setup...** from the menu bar.

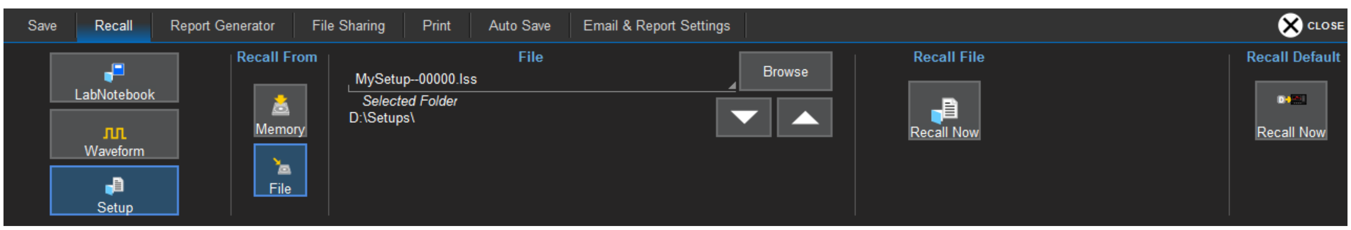
Recall Setup from Memory



1. Choose to Recall From **Memory**.
2. Touch one of the six **Recall** buttons under **Recall From Internal Setup...**.

Note: If a setup has been stored to a location, it is labeled with the save date/time. Otherwise, the slot is labeled **Empty**.

Recall Setup from File



1. Choose to Recall From **File**.
2. Enter the path to the **File**, or touch **Browse** and navigate to the desired file.

   If the file is in the Selected Folder, you can use the **Up /Down Arrows** to cycle through the available files until the desired file is selected.
3. Under Recall File, touch **Recall Now**.

Restore Default Setup

The front panel **Default Setup** button restores all the volatile setups to the factory default state.

You can also restore default settings via the touch screen:

1. Choose **File > Recall Setup** from the menu bar.
2. Choose to Recall From **File**.
3. Under Recall Default touch **Recall Now**.

LabNotebook

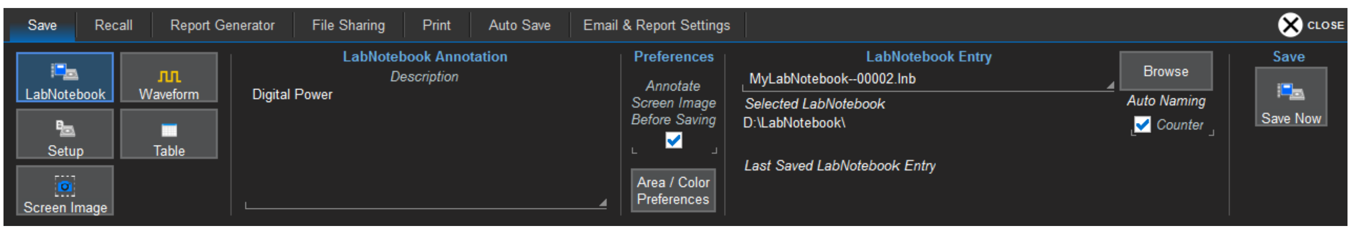
The LabNotebook feature allows you to create and save composite files containing a screen capture of all displayed waveforms, as well as all waveform and setup data at the time of capture. The Flashback feature instantly recalls the setups and waveforms stored with LabNotebook Entries, enabling you to restore the exact state of the instrument at a later date to perform additional analysis.

LabNotebooks can be output to a preformatted .PDF, .RTF, or .HTML report. You can also upload your own report layout if you prefer not to use the default.

Save LabNotebook

All LabNotebook files (.LNB) are composed of a screen image (.PNG), a setup file (LSS), and a waveform file (.TRC) for each waveform displayed. When creating LabNotebooks, you choose how to handle the screen image component, which is the basis for the Report Generator output, and to which you can add a description and other hand-drawn annotations.

Note: We strongly recommend that you stop acquisition before saving LabNotebooks, especially when running in Auto trigger mode. If you do not first stop, there is no guarantee that what you are capturing is what you want to save.



1. Choose **File > Save LabNotebook** from the menu bar.
2. Optionally, enter a **Description**. This text appears whenever the file is recalled and on reports.
3. To mark up the image using the drawing tools, select **Annotate Screen Image Before Saving**.

   To skip this step, deselect the checkbox. You can recall the LabNotebook later to add a description and annotations.
4. To change the area of the screen captured and the colors used, touch **Area/Color Preferences** and make your selections on the pop-up:
   - Use Print Colors (default) uses your print color palette (set in Preferences) on a white background to save ink. Deselect this to capture the display using a black background as it appears on the instrument.
   - Grid Area Only includes any visible waveform grids, tables, and descriptor boxes.
   - DSO Window includes the above plus any open dialogs, menu bar, and message bar.
   - Full Screen is the full Windows display, including other visible applications and wallpaper.

Touch **Close** to save your changes.

5. The system will auto name the file "MyLabNotebook" followed by a counter number. To change this to something more descriptive, enter the name in **LabNotebook Entry**, or touch **Browse** and use the File Browser. By default, LabNotebook files are saved to D:\LabNotebook on the instrument hard drive.

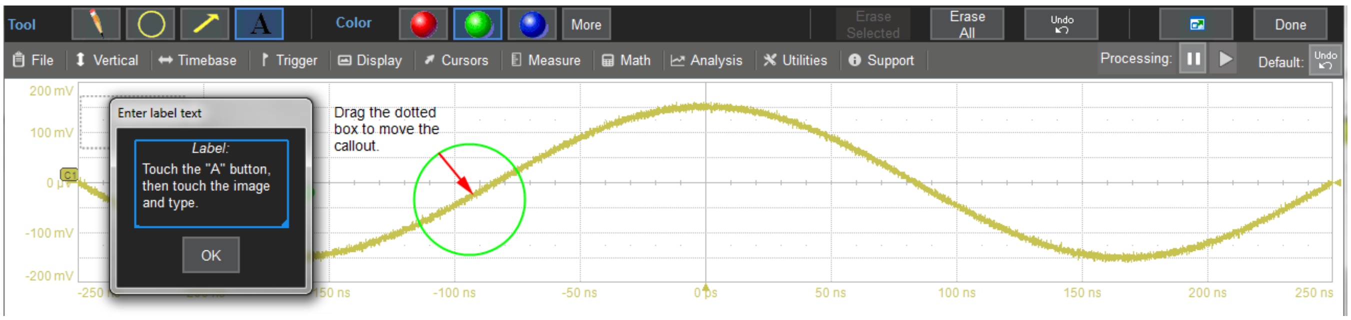
   Note: Changing the file path, name or format will reset the counter to the next number in that sequence.

6. If you do not want to use autonumbering to identify files, deselect **Counter**.
7. Touch **Save Now**.
8. If annotating, use the Drawing toolbar to mark up the screen image. Click **Done** when finished.

LabNotebook Drawing Toolbar

The basic LabNotebook is a screen capture of the display as it was at the time of entry, along with the setup and waveform data. If you have selected to Annotate Screen Image Before Saving, the capture is immediately displayed in the Drawing window. You can later add Annotations by recalling the LabNotebook into the Preview window and choosing to Annotate Screen Image.

Markup tools are available from the toolbar along the top of the window.



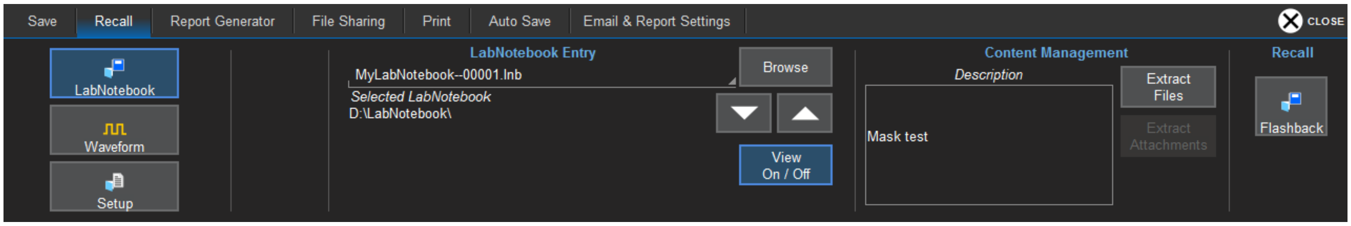
To use any tool, touch the icon, then touch on the image where you wish to draw or add text. From left to right, the tools are:

Tool	Function
Pencil	Draw in freehand. Maintain contact with the screen to make a continuous mark. Once you release, you can touch-and-drag the object to any point on the image.
Circle	Draw a circle around a waveform feature that you want to emphasize. Touch-and-drag across the diameter of the circle. When you release, the circle is placed. You can drag the circle to any location on the image.
Arrow	Draw lines with arrowheads for placing callouts. You can rotate these lines through 360 degrees or drag them to any location on the image.
Text	Open a textbox for placing labels/annotations on the image. Touch the point on the image to place the label, then enter the text in the pop-up dialog. Once placed, you can resize the textbox or drag it to any location on the image.
RGB Selectors	Quickly change the line color. Just touch the color icon, then choose the next drawing tool.
More	Activates a Custom line color field. The default color is Yellow. To choose another, touch the color swatch, then select from the Color dialog. You can enter RGB values, or choose from the spectrum. After saving, the new color appears in the Custom field. This remains the markup color until you choose another.
Erase / Erase All	Remove selected drawing objects. Erase All will also undo any Custom color selection.
Undo	Cancel the last action. Use it to restore any objects you inadvertently erased.
Move	Undock the drawing toolbar so you can move it anywhere on the display. This helps to keep tools handy when working on a particular area of a waveform. Touch the button again to restore the toolbar to the top of the Drawing window.
Done	Save the annotations with the image and close the Drawing window.

Recall LabNotebook

Once a LabNotebook Entry is saved, recall it to view and modify descriptive information or manage attachments and component files. Flashback to restore the oscilloscope to the state saved in the LabNotebook file.

Choose **File > Recall LabNotebook** from the menu bar. The last selected file will open in the Preview window.



Edit Description

You can modify the description and annotations saved with the LabNotebook Entry. Setups and waveform data originally saved with the LabNotebook cannot be changed.

1. Browse to and select the **LabNotebook Entry**.
2. Select **View On/Off** to preview the file.
3. From the top of the preview window, choose **Edit Description**.
4. Touch the pop-up description to open the virtual keyboard and edit the text.
5. Touch **OK** twice to save changes.

Annotate Screen Image

1. Browse to and select the **LabNotebook Entry**.
2. Select **View On/Off** to preview the file.
3. From the top of the preview window, choose **Annotate Screen Image** and use the drawing tools.
4. Click **Done** when finished to close the Drawing window and return to the preview.

Manage Attachments

To append external files to the LabNotebook composite (such as images of the DUT, mask files, test scripts, or anything relevant to the entry):

1. Browse to and select the **LabNotebook Entry**.
2. Select **View On/Off** to preview the file.
3. From the top of the preview window, choose **Manage Attachments** and select the files to attach. The file list will build in the lower part of the File Attachments browser.
4. Click **Update Attachments** to finish.

To later remove appended files, Manage Attachments and deselect the files from the list. Update

Attachments again to refresh the list.

Convert File Format

Individual entries in legacy .ZIP format Notebooks can be converted to the new LabNotebook file format.

1. In LabNotebook Entry, **Browse** to and select the legacy .zip file from the left side navigation pane of the File Browser.

   Note: Be sure to select the .zip file from the left pane so that its sub-entries appear in the right pane and the Save buttons appear on the browser. The file should appear in Current Path on the file browser.

2. To batch convert all entries into separate LabNotebook files, at the top of the file browser choose **Save All As LNB Files**.

   To convert a single entry, select it from the **Name** list (right side), then choose **Save As LNB File**.

New LabNotebooks of the same name as the original entries are created in the D:\XPort folder. These can be selected the same as any other .LNB files for Flashback, editing, reports, or extraction.

Extract Files

The component files that make up the LabNotebook composite (.PNG, .LSS, and .TRC) and any appended files can be extracted into separate files.

1. Enter the path to the **LabNotebook Entry**, or **Browse** to the file.
2. To extract all files, under Content Management, choose **Extract Files**.

   To extract only the attachments, choose **Extract Attachments**.

3. Navigate to the folder where you want the files extracted, and choose **Extract Now**.

   A folder of the same name as the original LabNotebook containing the separate files will be created at that location.

Flashback

Flashback restores the waveforms and setups saved with the LabNotebook Entry, so you can later analyze the inputs that resulted in that capture.

1. In **LabNotebook Entry**, enter the path to the LabNotebook, or touch **Browse** and navigate to the file.
2. Touch the **Flashback** button.
3. To exit Flashback, touch the **Undo** button at the far right of the menu bar.

Some result data *not* included in the Flashback are:

- **Persistence data**, although it is saved with the .LNB file and appears on reports.
- **Histogram data** over 16-bits.

- Floating point waveforms resulting from certain math operations that have much higher resolution than 16-bits. This extra resolution is not preserved when traces are recalled using Flashback.
- Cumulative Measurements in process when Flashback is entered. When Flashback is used, they lose their history and show instead only the results from the stored waveforms, not including any data taken from interim acquisitions.

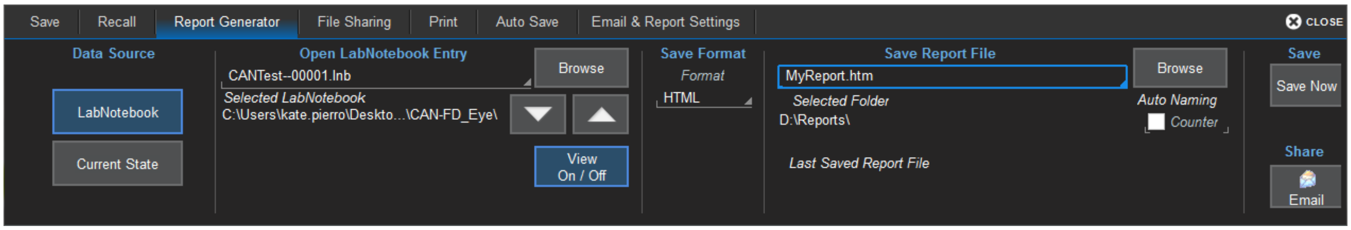
Report Generator

The Report Generator feature allows you to output a preformatted report consisting of an annotated screen image and a summary of the setups in place when it was captured. The report can be sourced from an existing LabNotebook, or it can be newly generated from the current state of the oscilloscope.

The logo and template used to create the report can be changed on the Email & Report Settings dialog.

Generate from LabNotebook

This procedure outputs an existing source as is. If you want to further annotate or change the description of a LabNotebook file, first recall the LabNotebook.



1. Choose **File > Report Generator** from the menu bar.
2. **Browse** to and select the **LabNotebook Entry**. If it is in the Selected Folder, just use the Up/Down Arrow keys to select it.
3. Optionally, use **View On/Off** to preview the selected file.
4. Choose a **Save Format** of HTML, .RTF, or .PDF.

   Note: Only HTML reports can later be saved to .RTF or .PDF format. The .RTF and .PDF reports can only be combined into another report of the same format.

5. By default, the system will auto name the file "Report" followed by a counter number. To change the file name or the Selected Folder, enter the full path and name in **Save Report File**, or touch **Browse** and use the File Browser. By default, report files are saved to the D:\Xport folder on the instrument hard drive.

   Note: Changing the file path, name, or format will cause the counter to reset to the next number in that sequence.

6. If you do not want to use autonumbering, deselect **Counter**.
7. To save the report, choose **Save Now**.

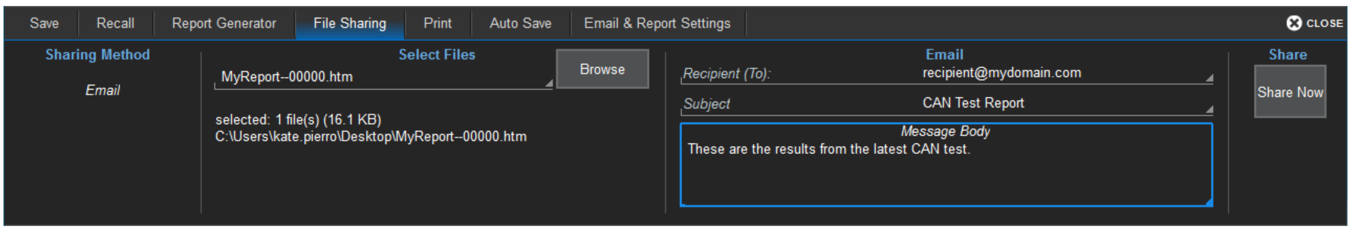
Generate from Current State

If you are creating a report from the current state of the instrument, the procedure is much like that above, except that you will also be able to enter a new **Description**, **Annotate Screen Image**, and select your capture **Area/Color Preferences** as you would normally do when first creating a LabNotebook. See Save LabNotebook for an explanation of these steps.

However, when using the Report Generator from Current State, you will not have saved a composite .LNB file that can later be recalled or extracted into its component files. You will only have the preformatted report file.

Share

Use the File Sharing dialog to email files from the instrument.



1. Choose **File > File Sharing** from the menu bar.
2. Use the File Browser to select all the files to be emailed.

   Use CTRL + Click to select multiple files.
3. Optionally, change the email **Recipient(s)**. This field defaults to whatever is in your Email settings.
4. Enter a **Subject** line and **Message Body**, then **Share Now**.

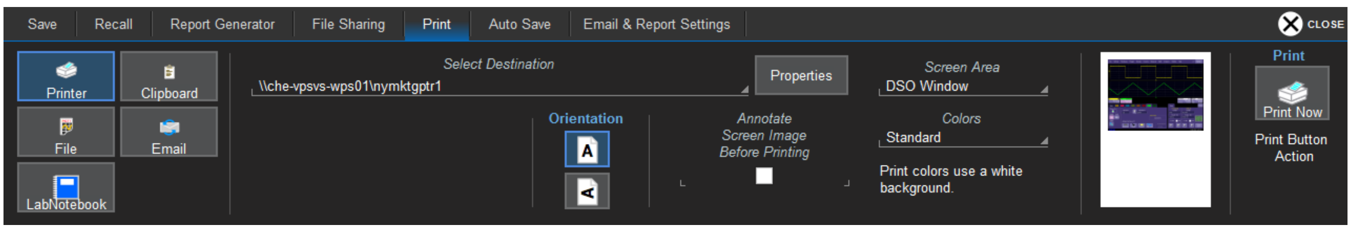
Print

Print dialog settings control how the oscilloscope's front panel Print button behaves.

Print captures an image of the touch screen display, but there are several options as to what it does next with the image:

- Send it to a **Printer** as formatted here.
- Copy it to the Windows **Clipboard**
- Save it to an image **File** using your current Screen Image Preferences
- **Email** an image file using your current Screen Image Preferences and Email Preferences
- Create a new **LabNotebook** file using your current LabNotebook Preferences

The **Print Now** button at the far right of the dialog can also be used to execute your current Print selection.



Printer Settings

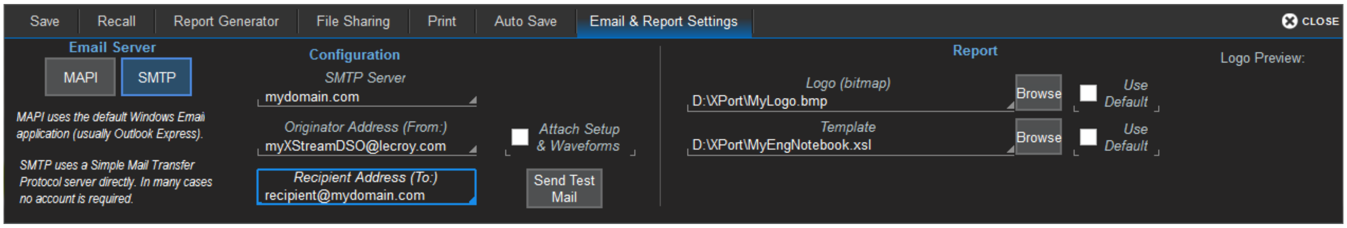
1. Touch **Select Destination** and choose a printer.
2. Touch **Properties** to change any of the printer properties.
3. Choose a paper orientation of **Portrait** or **Landscape**.
4. Select **Annotate Before Saving** to mark up captures before they print. Each time you press Print, the capture will open in the Drawing Tools window. When you are **Done** drawing, the image prints.

Capture Settings

1. Select a **Screen Area**:
   - Grid Area Only includes any visible waveform grids, tables, and descriptor boxes.
   - DSO Window includes the above plus any open dialogs, menu bar, and message bar.
   - Full Screen is the full Windows display, including other visible applications and wallpaper.
2. Choose the **Colors** used in the capture:
   - Standard uses a black background as it appears on the instrument.
   - Print uses your print color palette (set in Utilities > Preference Setup) on a white background
   - Black & White captures the image in grayscale.

Email & Report Settings

Configure the oscilloscope for email on the Email & Report Settings dialog.



Email Settings

1. Select to use **MAPI** or **SMTP**.
2. If you chose SMTP, touch **SMTP Server** and enter the network address of your mail server.
3. Touch **Originator Address (From:)** and enter the instrument's e-mail address.
4. Touch **Default Recipient Address (To:)** and enter the recipient's e-mail address.
5. Optionally, select to always **Attach Setup & Waveforms** associated with image files selected for email.
6. Use **Send Test Mail** to send a confirmation message to ensure proper e-mail configuration.

Report Settings

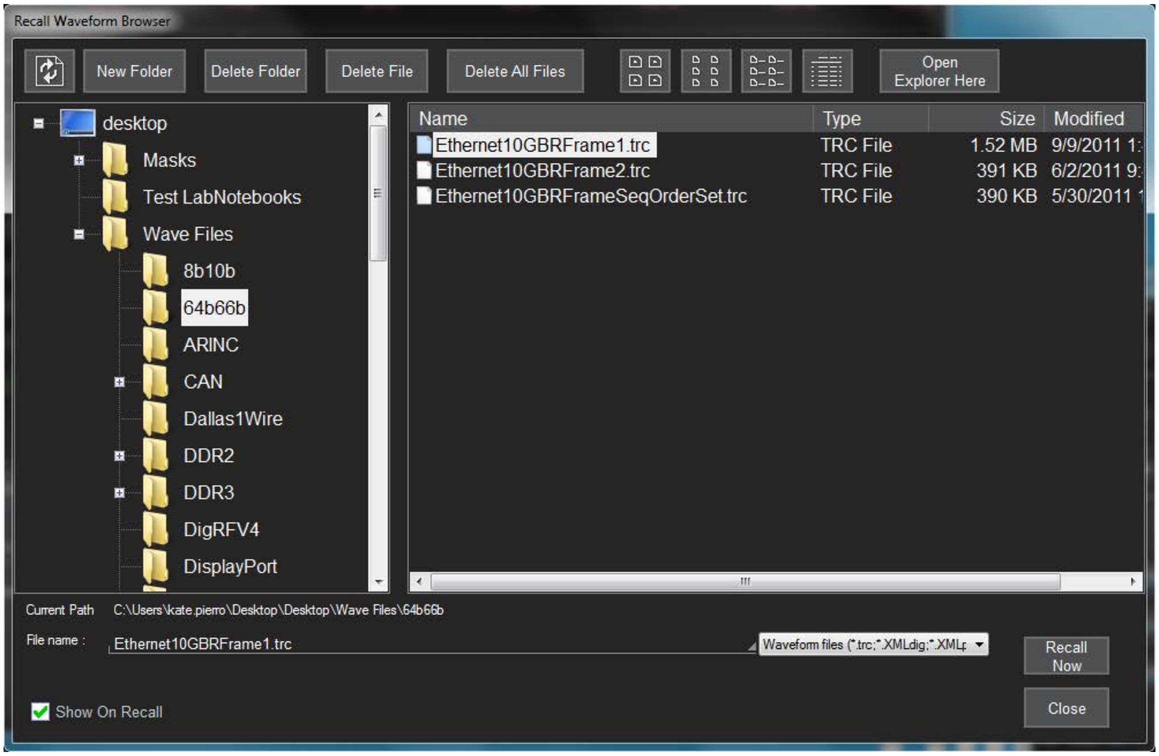
The default report template uses our logo as a placeholder. You can replace this with your logo. Logo files should be in bitmap (.bmp) format and not exceed 100 pixels high by 180 pixels wide. Place the file in the D:\Xport folder.

You can also use your own report template. Templates must be saved as .xsl or .xslt files and placed in the D:\Xport folder.

Deselect **Use Defualt** next to each item you want changed, then **Browse** to and select the new files.

Using the File Browser

Selecting **Browse** on any of the File menu dialogs opens a File Browser that enables you to perform basic disk utility functions, as well as making file path/name and format selections.



Disk Utilities

New Folder, **Delete Folder**, **Delete File**, and **Delete All Files** can be used to change your file system prior to saving new files. Be sure to first select the folder from the navigator (left) and file from the file list (right).

The buttons immediately above the file list let you change how items appear in the browser: icons only, details, etc.

File Path and Name

Use the navigator pane to the right to browse directories on the internal hard drive or USB drive. To select an existing file to copy, move, change name, etc, use the file list to the right of the navigator. When an editable file is selected, the **Current Path** and **File Name** are shown immediately below the navigator pane.

If you do not see the file you seek, try using the drop-down next to **File name** to change the file type to **"All files (*.*)"**.

Auto Naming Selections

The checkboxes to turn on/off the **Source** prefix and the **Counter** number suffix used to autogenerate file names will appear on the File Browser when saving files. These selections are linked to those on the underlying dialog, and changing the value in either place causes it to change everywhere.

Note: If you change any part of the file path/name or format on the File Browser or the dialog, the Counter number will reset to the last number in the sequence associated with that value.

Example: Changing "Decode1-Table-0002" to "Decode1-CANFDTable-..." will reset the counter to "Decode1-CANFDTable-0000" if there is no prior file named "Decode1-CANFDTable-..." .

Actions

The **Close** button accepts the selections you made on the File Browser and closes the browser window. If you do not need to make further entries on the dialog to complete your task, you can choose to Save, Recall, Flashback, Email, etc. right from the File Browser.

Utilities

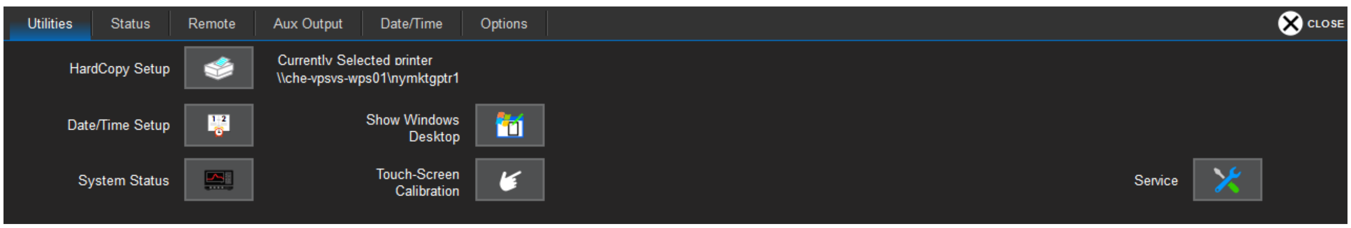
Utilities settings primarily control the instrument's interaction with other devices/systems.

Preferences settings, on the other hand, tend to control the appearance and performance of the oscilloscope application. Use these settings to personalize the behavior of the instrument.

Disk Utilities enable you to perform commonplace Windows file functions.

Utilities Dialog

To access the Utilities dialogs, choose **Utilities > Utilities Setup...** from the menu bar.



User Action, Date/Time Setup and System Status buttons open their corresponding dialogs.

Show Windows Desktop minimizes the oscilloscope application window. Maximize the application by touching the display icon located at the lower-right of the desktop.

Touch-Screen Calibration launches a sequence of display calibration screens. You will be prompted through a series of actions to improve the precision and accuracy of the touch screen.

The **Service** button to the far right of the dialog launches a section of the application reserved for qualified Teledyne LeCroy service personnel. An access code is required to enter this section.

Status

The Utilities Status dialog displays information about your instrument including model number, serial number, firmware version, and installed hardware and software options.

Choose **Utilities > Utilities Setup** from the menu bar, then touch the **Status tab**.

Or

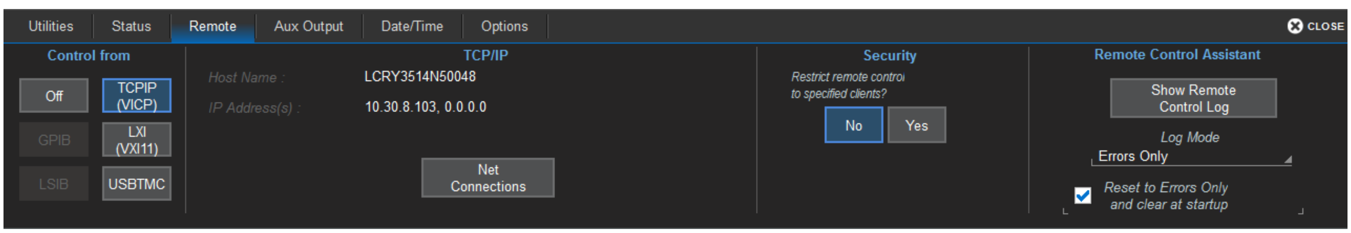
Choose **Support > About** from the menu bar.

Remote Control

The Remote dialog contains settings to configure remote control of the instrument and also network access. Supported remote control protocols are:

- **TCPIP (Ethernet)**. If you choose this option, also install Teledyne LeCroy's VICP drivers on the controller. These are included in the VICP Passport plug-in, available free from the teledynelecroy.com software download page under Oscilloscope Downloads > Software Utilities. The instrument uses Dynamic Host Configuration Protocol (DHCP) as its default addressing protocol. You can assign a static IP address using the standard Windows network setup menus.
- **LXI (Ethernet)**. To use this option on Windows 10 oscilloscopes, you must run the oscilloscope from the administrative user account, LCRYADMIN. See Switching Windows User.
- **USBTMC**
- **GPIB**. This selection is active if you are using the USB2-GPIB adapter. Connect the adapter from any USB port on the oscilloscope to the GPIB port on the controller.

Note: See the *MAUI Oscilloscopes Remote Control and Automation Manual* for full instructions on making the remote connection and sending remote commands. The steps shown here are only those performed on the oscilloscope to enable the connection. A commented copy of the oscilloscope Waveform Template (.tpl) file is installed on the oscilloscope in C:\Program Files\LeCroy\XStream. You can open this ASCII file using any text editor, such as Windows Notepad, which is also installed on the oscilloscope. The template will help you better understand the MAUI architecture for transferring waveform data to and from the oscilloscope.



Set Up Remote Control

1. From the menu bar, choose **Utilities > Utilities Setup**, then touch the **Remote tab**.
2. On the **Remote** dialog, make a **Control From** selection.
3. If using TCPIP and wish to restrict control to specific network clients, choose **Yes** under Security. Enter the IP addresses or DNS names of the authorized controllers in a comma-delimited list.

Remote Control Assistant Event Log

The Remote Control Assistant monitors communication between the controller and instrument. You can log all transfers or errors only. The log can be output to an ASCII file, valuable when creating and debugging remote control programs.

1. From the menu bar, choose **Utilities > Utilities Setup**, then touch the **Remote tab**.
2. Under **Remote Control Assistant**, touch **Log Mode** and choose **Off**, **Errors Only**, or **Full Dialog**.
3. To always clear the log at startup, check **Reset to Errors Only and clear at startup**.

Export Contents of the Event Log

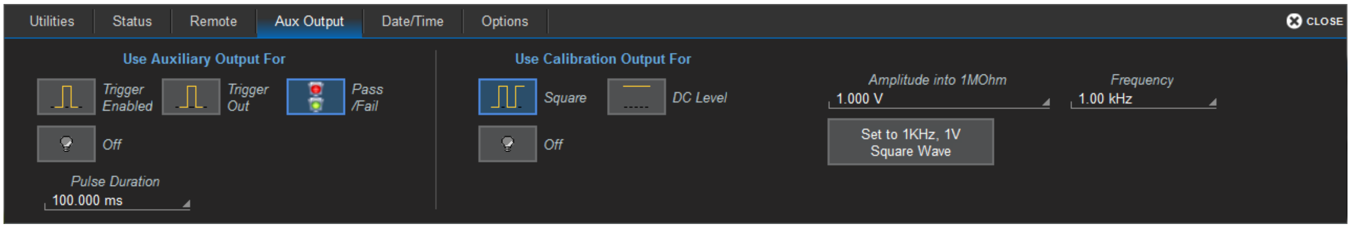
1. From the menu bar, choose **Utilities > Utilities Setup**, then touch the **Remote tab**.
2. Touch the **Show Remote Control Log** button. The Event Logs pop-up is shown.
3. Enter a log file name in **DestFilename**, or touch Browse and navigate to an existing file.

   Note: New contents will overwrite the existing content; it is not appended.

4. Touch **Export to Text File**.

Auxiliary Output

Use the Aux Output dialog to configure the output of the **Aux Out** port and **Cal Out** port.



Auxiliary Output

The Aux Out port outputs a 3.3 V CMOS into high impedance pulse following the selected event:

- **Trigger Enabled** sends a pulse when the trigger is ready (Ready indicator lit), but not necessarily triggered. It can be used as a gating function to trigger another instrument.
- **Trigger Out** sends a pulse upon a trigger (TRIG'D Indicator lit). Enter the desired **Pulse Duration**.
- **Pass/Fail** sends a pulse when Pass/Fail test conditions are met. Enter the desired **Pulse Duration**. Be sure to also select Pulse Aux Out on the Pass/Fail Actions dialog.

Off disables output.

Calibration Output

On models equipped with a Cal Out hook on the front of the instrument, use these settings to configure the output signal :

- For a **Square** wave signal, enter the wave **Frequency** and **Amplitude into 1 MΩ**, or choose to **Set to 1 kHz, 1 V Square Wave**.
- For a reference **DC Level**, enter an **Amplitude into 1 MΩ**.

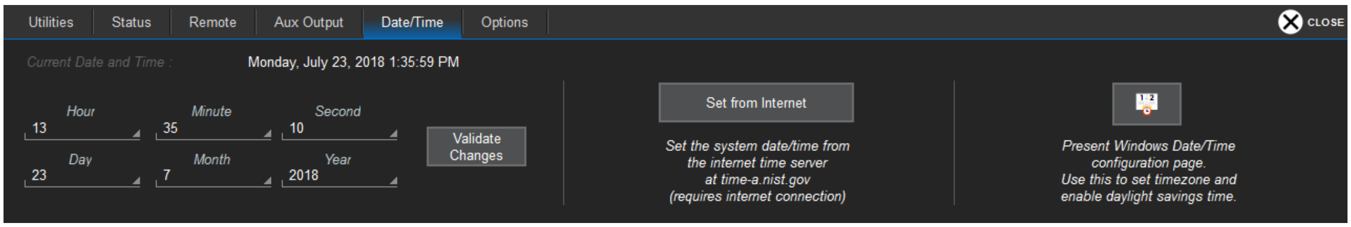
Off disables output.

Date/Time

Date/Time settings control the instrument's timestamp. These numbers appear in the message bar and on tables/records internal to the oscilloscope application, such as History Mode and WaveScan.

Note: On Windows 10 oscilloscopes, first switch to the Admin User LCRYADMIN to change the Date/Time settings.

To access the Date/Time dialog, choose **Utilities > Utilities Setup** from the menu bar, then touch the **Date/Time tab**.



Manual Method

Enter the **Hour**, **Minute**, **Second**, **Day**, **Month**, and **Year**, then touch the **Validate Changes** button.

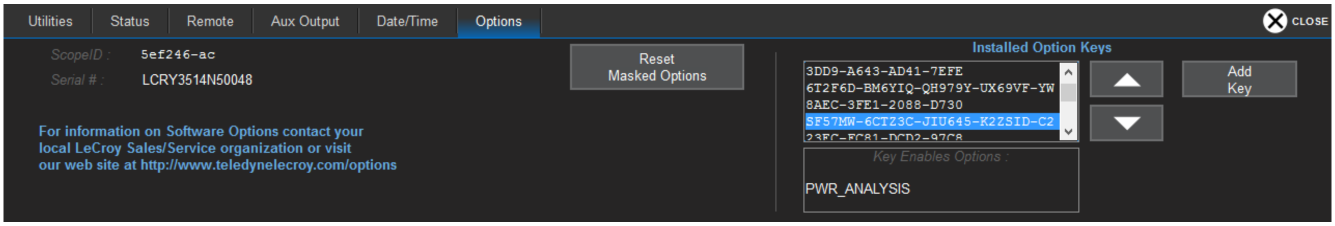
Internet Method

This method uses the Simple Network Time Protocol (SNTP) to read the time from time-a.nist.gov. The instrument must be connected to an internet access device through a LAN (Ethernet) port.

If your connection is active, touch the **Set from Internet** button.

Options

Many optional software packages are available to extend the Analysis functions of the instrument. When you purchase an option, you will receive a key code by email that activates the new functionality. Use the **Options** dialog to activate software options by installing the key code. This dialog also displays the **ScopeID** and **Serial #**.



To install a key:

1. From the menu bar, choose **Utillities > Utilities Setup**, then open the **Options tab**.
2. Touch **Add Key**. The Virtual Keyboard appears onscreen.
3. Use the Virtual Keyboard to type the Key Code in the **Enter Option Key** field, then touch **OK** to enter the information.

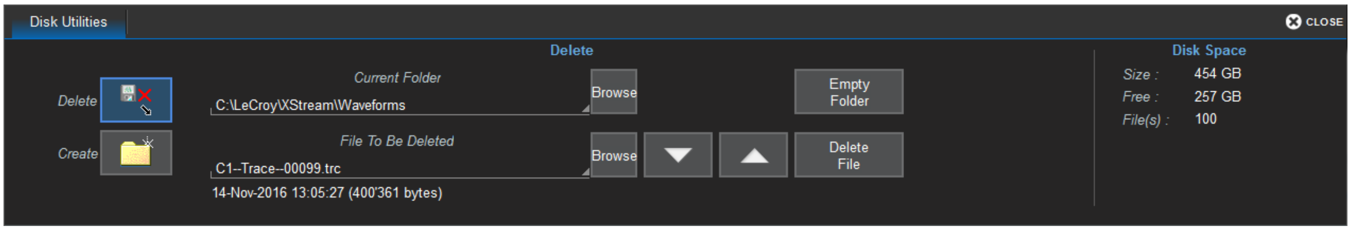
   The Key Code is added to the list of Installed Option Keys. You can use the Up/Down buttons to scroll the list. The software option that each key activates is displayed below the list.
4. Choose **Yes** to restart the oscilloscope application.

Disk Utilities

Use the Disk Utilities dialog to manage files and folders on your instrument's hard drive. Disk Space information is shown at the far right of the dialog for convenience.

Note: These tasks can also be accomplished using the standard Microsoft Windows file management tools. Choose **File > Minimize** to access the Windows desktop and task bar.

Access the **Disk Utilities** dialog by selecting **Utilities > Disk Utilities** from the menu bar.



Delete a Single File

1. Touch the **Delete** button.
2. **Browse** to the current folder containing the file.
3. **Browse** to the file to be deleted, or use the **Up** and **Down** arrow buttons to scroll through the files in the folder.
4. With the desired file selected, touch **Delete File**.

Delete All Files in a Folder

1. Touch the **Delete** button.
2. **Browse** to the current folder containing the file.
3. With the desired folder selected, touch **Empty Folder**.

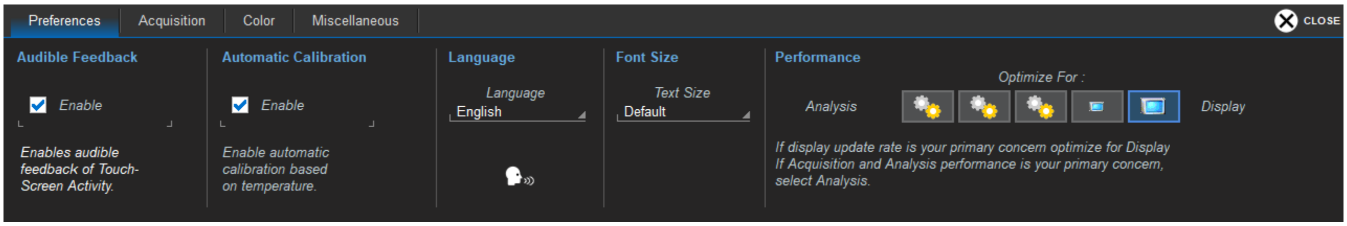
Create a New Folder

1. Touch **Create**.
2. Touch **Current folder** and provide the full path to the new folder, including the folder name.
3. Touch **Create Folder**.

Preferences Dialogs

Preference settings have mostly to do with the appearance and performance of the instrument itself, rather than its interaction with other devices/systems. These settings are called "non-volatile," because they are not lost when the oscilloscope is restarted and do not change when a setup panel is recalled.

Access the Preferences dialogs by choosing **Utilities > Preference Setup...** from the menu bar.



Power on AC will turn on the oscilloscope and launch the software as soon as you connect to AC power, without having to first press the Power button.

Audible Feedback controls the instrument's audio output. Select this box to hear a beep each time you touch a screen or front panel control.

Language sets the language used on the touch screen display.

Font Size (Not available on Windows 10 and Windows CE oscilloscopes) changes the size of the text that appears on the touch screen display.

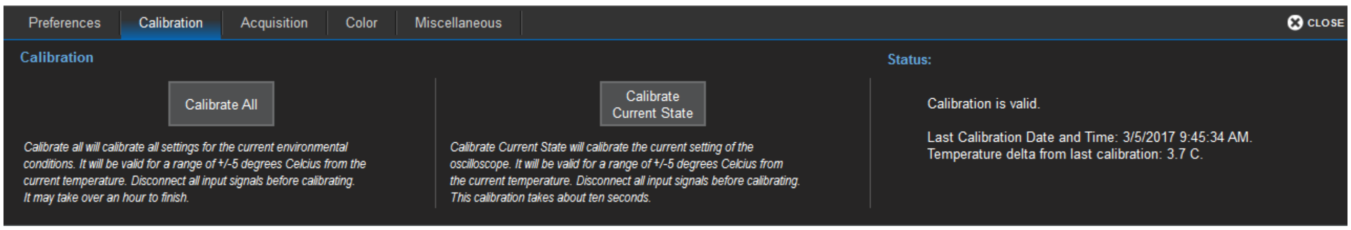
Tip: To change font size and other screen settings on Windows 10 oscilloscopes, swipe from the right of the touch screen to display the Action Center, then choose All Settings > System Settings and make your selections.

Performance settings let you optimize performance for either **Analysis** (speed of acquisition and calculation) or **Display** (speed of update/refresh). For example, if you are concerned with persistence or averaging, you might optimize for Analysis, giving higher priority to waveform acquisition at the expense of display update rate. Choices are presented as a spectrum.

Calibration

Calibration ensures that the output from the analog-to-digital converters (ADCs) accurately represents the input. The oscilloscope is calibrated at the factory at 23 °C (± 2 °C) prior to shipment. So that it maintains specified performance, it is factory set to perform an automatic calibration routine upon power up.

Note: Warm the oscilloscope for at least 20 min. after power on to ensure it reaches a stable operating temperature and completes the calibration routine. You will see a warning message on the Calibration dialog when the oscilloscope is still in the warm-up phase. Specifications are not guaranteed during warm up.



High Definition oscilloscopes utilize Temperature Dependent Calibration. Within ±5 °C of the original calibration temperature, the oscilloscope should meet all specifications once warmed up.

We recommend manual calibration if using the oscilloscope in an environment more than ± 5 °C from the calibration temperature, or if it has been over one month since the previous calibration. From the menu bar, choose **Utilities > Preference Setup > Calibration**. The Status section of the Calibration dialog will tell you if calibration is needed for the current vertical and horizontal settings at the current environmental conditions. There are two routines available for selection:

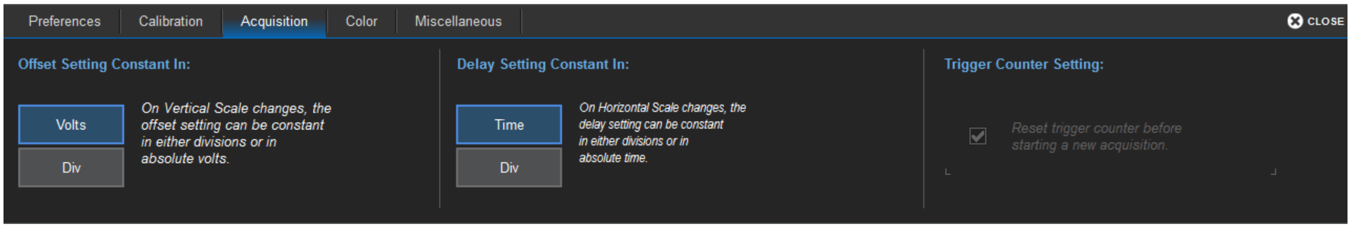
- **Calibrate All** calibrates all possible combinations of vertical and horizontal settings at the current environmental conditions. This calibration is valid for the current temperature ±5 °C and may take over an hour.
- **Calibrate Current Setting** calibrates the current vertical and horizontal settings at the current environmental conditions. This calibration is valid *for these settings only* at the current temperature ±5 °C and takes about 10 seconds.

To maintain good performance, we recommend that you **Calibrate All** about every month.

Caution: Remove all inputs prior to calibrating the oscilloscope.

Acquisition

The Acquisition preference settings determine how traces behave as Vertical Offset or Horizontal Delay changes. Choose **Utilities > Preference Setup** to open the **Acquisition** dialog.



Offset Setting constant in:

- **Volts** keeps the amount of Offset in the amount of Volts specified, regardless of the V/div setting. As the Offset is adjusted, the trace will appear to move up or down relative to the zero level.
- **Div**(isions) keeps the Vertical Offset level indicator stationary. The waveform remains on the grid as you change V/div, but your Offset value will change.

Delay Setting constant in: (Not available on WaveRunner 8000HD or WavePro HD)

- **Time** keep the Horizontal Delay at the amount of time specified, regardless of the Time/div setting. The Horizontal Delay indicator moves relative to the 0 point at the center of the grid.
- **Div**(isions) keeps the Horizontal Delay indicator stationary. The trigger point remains on the grid as you increase the timebase; whereas, if Time is selected, the trigger point could move off the grid.

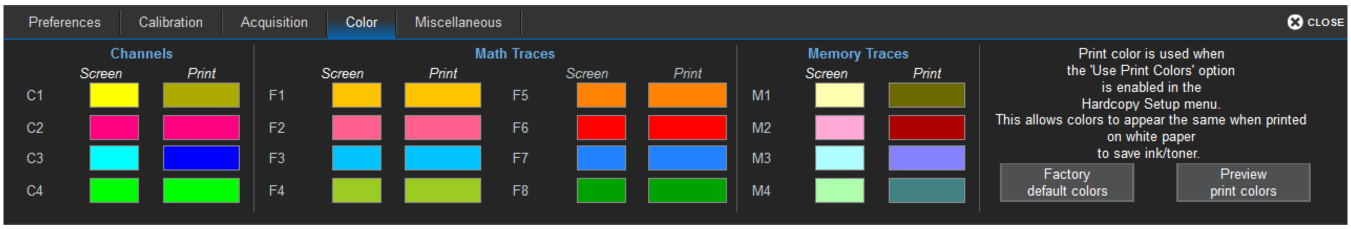
Note: Whenever Div is selected, values are scaled proportional to the change in gain or timebase, thereby keeping the division of the grid constant.

Trigger Counter Setting is selected by default. It clears the trigger counter each time a new acquisition command is sent. It is only made active for deselection when trigger Holdoff is set.

Color

Color dialog settings assign the colors used for channel, math, and memory traces. All dialogs, tables, and trace descriptor boxes will match the color of the trace assigned here. You can choose different colors to be used on the instrument and in print.

Note: Print colors are used only when the Colors control is set to Print on any of the File menu dialogs (Save Screen Image, Print, etc.). Otherwise, the Screen colors are used for print output as well as on screen.



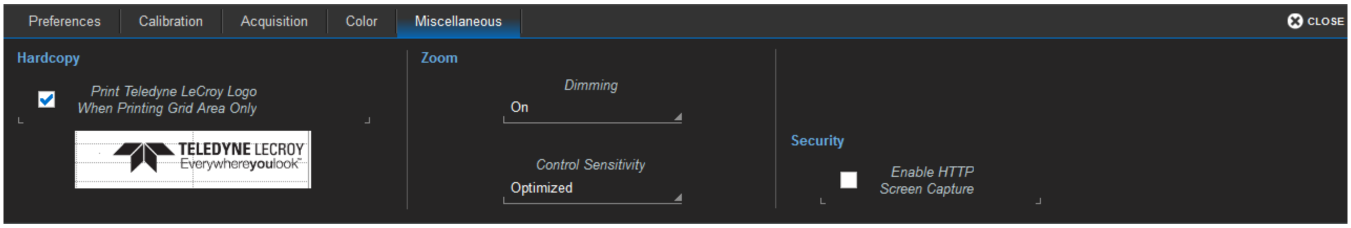
To make any setting, just touch the color swatch for either Screen or Print next to the trace number, and make a selection from the Color pop-up menu.

For convenience, you can **Preview print colors** to see how the settings will appear in print output.

Touch **Factory default colors** to recall the original color settings for your instrument.

Miscellaneous

These other Preference settings are located on the **Miscellaneous** dialog.



To add the Teledyne LeCroy logo to print output, check **Print Teledyne LeCroy Logo When Printing Grid Area Only**. This identifies the instrument as the source of the image.

Dimming darkens/shades those areas of the source waveform that are not part of the Zoom trace.

Control Sensitivity adjusts the sensitivity of the front panel knobs. **Optimized** applies an acceleration algorithm to the knobs. **Legacy** detects rotation of the front panel knobs in a manner similar to our legacy oscilloscopes.

Serial Decode Annotation Position: If you have Serial Trigger or Decode options installed on your instrument, this control determines the placement of annotation labels relative to the trace line. It does not appear if there are no installed options.

- **On Trace** places the label close to the line.
- **On Noisy Trace** sets the label further away to accommodate potential noise spikes in the trace.

Check **Enable HTTP Screen Capture** to enable remote capture of the touch screen display over a netowrk. This setting is required to use the instrument with the WaveStudio software.

Maintenance

Topics in this section describe procedures for keeping the instrument in optimal working condition.

Restart/Reboot Instrument

To restart the oscilloscope application, choose **File > Exit** from the menu bar, then touch the **Start DSO** desktop shortcut.

To reboot the instrument, which includes restarting the Windows OS:

1. From within the oscilloscope application, choose **File > Shutdown**.
2. Wait 10 seconds after the oscilloscope has fully shut down, then press the **Standby Power button** on the front of the instrument.

Restore Default Setup

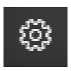
The front panel **Default Setup** button restores all the volatile setups to the factory default state.

You can also restore default settings via the touch screen:

1. Choose **File > Recall Setup** from the menu bar.
2. Choose to Recall From **File**.
3. Under Recall Default touch **Recall Now**.

Changing Screen Settings

On instruments running Windows 10, MAUI software font size and other screen settings are now changed using the standard Windows 10 controls, rather than through the MAUI application. The text font is set to display at 125% by default.

1. Swipe the touch screen from the right to display the Action Center.
2. Touch the **All Settings**  icon .
3. Choose **System** settings.
4. Make your selections from the Settings dialog.

   Tip: If prompted, enter **password SCOPEADMIN** to permit changes.

5. Swipe from the right again to return to MAUI.
6. Choose **File > Shutdown**, then press the Power button to reboot.

Touch Screen Calibration

Periodically calibrate the touch screen to maintain its accuracy and responsiveness. We recommend that you use a stylus rather than your finger for this procedure.

1. From the menu bar, choose **Utilities > Utilities Setup**.
2. On the Utilities main dialog, touch **Touch-Screen Calibration**.
3. Following the prompts, touch as close as possible to the center of each cross that appears on the screen until the calibration sequence is complete.

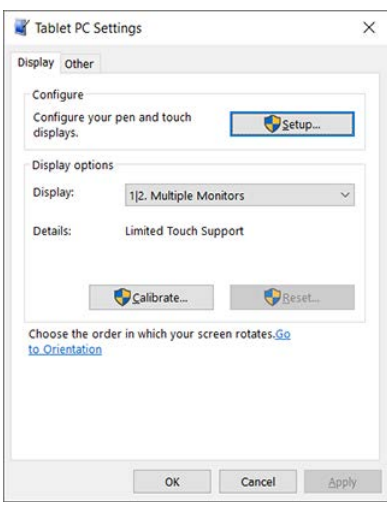
Windows 10 External Display Setup

The Windows 10 default is to treat any external monitor connected to a Windows 10 PC as the primary touch screen. On Windows 10 oscilloscopes with an external monitor attached, you will have to manually change the display settings so that the oscilloscope is recognized as a touch screen.

If the external monitor is touch screen enabled and you wish to use it along with the oscilloscope to control the oscilloscope application, you will need to repeat this procedure to also identify it as a touch screen.

1. Follow manufacturer's instructions to connect your monitor to the oscilloscope.
2. Choose **File > Minimize** to display the Windows Desktop.
3. Choose **Start** and enter the **Search** term "**Tablet PC Setup**."

   You should see the Tablet PC Settings dialog.



4. If the external monitor is *not* touch screen enabled, the Display selection will be 1|2 Multiple Monitors. This is OK.

   If the external monitor *is* touch screen enabled, be sure **Display 1** (the oscilloscope) is selected.
5. Touch the **Setup** button.

6. When prompted for permissions, enter the password **SCOPEADMIN** and touch **Yes**.

   The oscilloscope display will turn white, with the text "Touch this screen to identify it as the touch screen. If this is not the Tablet PC screen, press Enter to move to the next screen."

7. Touch the oscilloscope display.

   You will see the instruction: "Press Enter to proceed to the next step to complete your configuration."

8. Press **Enter**.

9. If you wish to also configure the external display as a touch screen, repeat this procedure selecting the external monitor name from the Display field on the Tablet PC Setup dialog.

Software and File Management

Due to the increased security of Windows 10, many more operations require Administrator permissions than did with previous versions of Windows. When doing any of the following from the default LeCroyUser account on Windows 10 oscilloscopes, you will be asked to supply the administrative **password SCOPEADMIN** (all uppercase).

- **Installing software**, both third-party applications and MAUI updates
- **Accessing third-party applications** or running an application as an administrator
- **Admitting third-party applications** to make changes to the MAUI application
- **Using Device Manager** and making changes within it (such as installing new device drivers)
- **Moving, deleting, or copying certain files**
- **Changing global settings**, such as Date/Time

Note: Certain QualiPHY compliance packages require you to run as an Administrator. See the product documentation for an indication of when this is required.

Switching Windows Users

To maintain the highest security profile, the oscilloscope is shipped to run as the Standard user, LeCroyUser. **If you need to run as an Administrator, do the following to switch to LCRYADMIN.** This will put the oscilloscope into a "legacy" mode where it will run most like it did on Windows 7.

1. Choose **File > Exit** to exit the oscilloscope application.
2. Open the Windows **Start menu**.
3. Hover over the Teledyne LeCroy logo and select **user LCRYADMIN**.
4. Enter the password **SCOPEADMIN** (all uppercase).

To return to the default LeCroyUser account, repeat this procedure selecting **LeCroyUser** and entering the password **lecroyservice** (all lowercase).

You may create as many other new user accounts on the oscilloscope as you wish, provided you are logged in as LCRYADMIN when doing so. Contact your local Teledyne LeCroy office for instructions on creating new users without disrupting oscilloscope performance.

Note: As long as there are *any* Standard (non-administrative) users, the oscilloscope will reboot into the last active Standard user, regardless of whether you've switched to an administrative user. The only way to change this is to make LeCroyUser (and any other new users you create) an Administrator. Contact Customer Support for instructions on changing user privileges and creating new users on the oscilloscope.

MAUI Firmware Update

The MAUI™ Software Setup Wizard can be used to install Teledyne LeCroy desktop software or oscilloscope firmware, including the MAUI application, required DLLs, device drivers and low-level microcode for integrated circuits. Follow these instructions for an oscilloscope firmware update.

The update *does not* modify or delete any saved panel setups, waveforms, screen captures, calibration constants, or other data stored on the D: drive.

Caution: Do not install any firmware version prior to 8.6.1.8 on Windows 10 oscilloscopes. Doing so will disrupt the normal behavior of the software, unless you run the recovery procedure. To install firmware on Windows 10 machines, you must be logged on as an Administrator or supply the password SCOPEADMIN. The installation may take several minutes. **Do not power down at any point during the process.**

Download Instructions

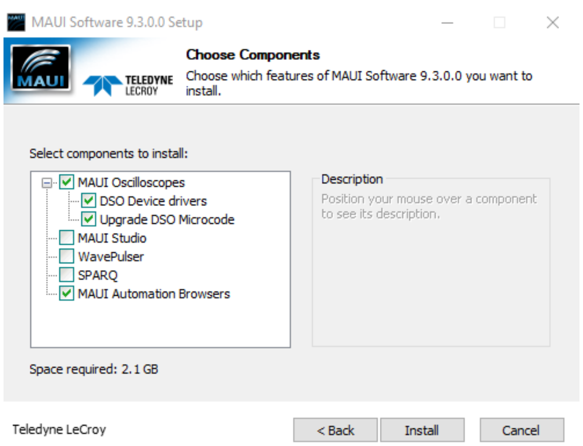
Tip: If the oscilloscope has an internet connection, choose File > Exit to close MAUI and use the oscilloscope browser to download the installer directly.

1. Visit the software download page at teledynelecroy.com and go to **Oscilloscope Downloads > Firmware Upgrades**.
2. Select your **series** and **model number**.
3. Enter your registration **login** information, or create a new account. You cannot proceed without an account.
4. Click the **download** link, and choose to **Save** the installer to the oscilloscope Desktop or to a USB storage device for transfer to the instrument.

Installation Instructions

1. From the oscilloscope menu bar, choose **File > Exit** to close MAUI.
2. Browse to **MAUI Installer64_<*version*>.exe** and double-click it to launch the setup wizard.
3. On Windows 10 instruments running as a Standard user (the default state), enter the **Password SCOPEADMIN** (all uppercase).
4. Click **Next** to begin installation.
5. Review and accept the license agreement. Click **Next**.
6. At the Choose Components dialog, select the checkboxes for:
   - **MAUI Oscilloscopes**, including **DSO Device Drivers** and **Upgrade DSO Microcode**
   - **MAUI Automation Browsers**

   Optionally, select SPARQ if you wish to drive a connected SPARQ from the instrument.



7. Click **Install**.
8. If you receive Windows security warnings, **trust** and **Install** the file.
9. If you see the Hardware Programmers screen, accept all code installations, then click the **closebox** *after* you see that programming has completed to return to the setup wizard.
10. When installation is complete, choose **Reboot now** and click **Finish**.

Note to QualiPHY Users

MAUI and QualiPHY software versions must match to a point-release to function properly. If you are upgrading your oscilloscope firmware, upgrade your version of QualiPHY as soon as possible.

Technical Support

Live Support

Registered users can contact their local Teledyne LeCroy service center at the number listed on our website.

You can also submit Technical Support requests via the website at:

teledynelecroy.com/support/techhelp

Resources

Teledyne LeCroy publishes a free Technical Library on its website. Manuals, tutorials, application notes, white papers, and videos are available to help you get the most out of your Teledyne LeCroy products. Visit:

teledynelecroy.com/support/techlib

The Datasheet published on the product page contains the detailed product specifications.

Service Centers

For a complete list of offices by country, including our sales & distribution partners, visit:

teledynelecroy.com/support/contact

Teledyne LeCroy
700 Chestnut Ridge Road
Chestnut Ridge, NY, 10977, USA
teledynelecroy.com

Sales and Service:
Ph: 800-553-2769 / 845-425-2000
FAX: 845-578-5985
contact.corp@teledynelecroy.com

Support:
Ph: 800-553-2769
customersupport@teledynelecroy.com

Returning a Product for Service

Contact your local Teledyne LeCroy service center for calibration or other service. If the product cannot be serviced on location, the service center will give you a **Return Material Authorization (RMA) code** and instruct you where to ship the product. All products returned to the factory must have an RMA.

Return shipments must be prepaid. Teledyne LeCroy cannot accept COD or Collect shipments. We recommend air-freighting. Insure the item you're returning for at least the replacement cost.

1. Remove all accessories from the instrument.
2. Label the instrument with:
   - The RMA
   - Name and address of the owner
   - Description of failure or requisite service
3. Pack the instrument in its original shipping box, or an equivalent carton with adequate padding to avoid damage in transit. Do not include the manual.
4. Mark the outside of the box with the shipping address given to you by Teledyne LeCroy. Be sure to add the following:
   - ATTN: *<RMA code assigned by Teledyne LeCroy>*
   - FRAGILE
5. **If returning a product to a different country:** contact Teledyne LeCroy Service for instructions on completing your import/export documents.

Extended warranty, calibration, and upgrade plans are available for purchase. Contact your Teledyne LeCroy sales representative to purchase a service plan.

For a complete list of Teledyne LeCroy offices by country, including our sales and distribution partners, visit: **teledynelecroy.com/support/contact**

Teledyne LeCroy
700 Chestnut Ridge Road
Chestnut Ridge, NY, 10977, USA

Sales and Service:

Ph: 800-553-2769 / 845-425-2000
FAX: 845-578-5985
contact.corp@teledynelecroy.com

Support:

Ph: 800-553-2769
customersupport@teledynelecroy.com

Certifications

Teledyne LeCroy certifies compliance to the following standards as of the time of publication. See the *EC Declaration of Conformity* document shipped with your product for the current certifications.

EMC Compliance

EC Declaration of Conformity- EMC

The instrument meets the intent of EC Directive 2014/30/EU for Electromagnetic Compatibility. Compliance was demonstrated to the following specifications listed in the Official Journal of the European Communities:

EN 61326-1:2013 and EN 61326-2-1:2013, EMC requirements for electrical equipment for measurement, control, and laboratory use. [1]

Electromagnetic Emissions:

EN 55011:2016+A1:2017, Radiated and Conducted Emissions Group 1, Class A [2] [3]

EN 61000-3-2:2014, Harmonic Current Emissions, Class A

EN 61000-3-3:2013, Voltage Fluctuations and Flickers, Pst = 1

Electromagnetic Immunity:

EN 61000-4-2:2009, Electrostatic Discharge, 4 kV contact, 8 kV air, 4 kV vertical/horizontal coupling planes [4]

EN 61000-4-3:2006+A2:2010, RF Radiated Electromagnetic Field, 3 V/m 80-1000 MHz; 3 V/m 1400 MHz-2 GHz; 1 V/m 2 GHz-2.7 GHz

EN 61000-4-4:2012, Electrical Fast Transient/Burst, 1 kV on power supply lines, 0.5 kV on I/O signal data and control lines [4]

EN 61000-4-5:2014+A1:2017, Power Line Surge, 1 kV AC Mains L-N, L-PE, N-PE [4]

EN 61000-4-6:2014, RF Conducted Electromagnetic Field, 3 Vrms 0.15 MHz-80 MHz

EN 61000-4-11:2004+A1:2017, Mains Dips and Interruptions, 0%/1 cycle, 70%/25 cycles, 0%/250 cycles [4] [5]

1. To ensure compliance with all applicable EMC standards, use high quality shielded interface cables.
2. Emissions which exceed the levels required by this standard may occur when the instrument is connected to a test object.
3. This product is intended for use in nonresidential areas only. Use in residential areas may cause electromagnetic interference.
4. Meets Performance Criteria “B” limits of the respective standard: during the disturbance, product undergoes a temporary degradation or loss of function or performance which is self-recoverable.
5. Performance Criteria “C” applied for 70%/25 cycle voltage dips and 0%/250 cycle voltage interruption test levels per EN61000-4-11.

*European Contact:**

Teledyne GmbH
Im Breitspiel 11c
D-69126 Heidelberg
Germany
Tel: + 49 6221 82700

Australia & New Zealand Declaration of Conformity– EMC

The instrument complies with the EMC provision of the Radio Communications Act per the following standards, in accordance with requirements imposed by Australian Communication and Media Authority:

AS/NZS CISPR 11:2015, Radiated and Conducted Emissions, Group 1, Class A

AUSTRALIA / NEW ZEALAND CONTACTS:*

RS Components Pty Ltd.
Suite 326 The Parade West
Kent Town, South Australia 5067

RS Components Ltd.
Unit 30 & 31 Warehouse World
761 Great South Road
Penrose, Auckland, New Zealand

*Visit teledynelecroy.com/support/contact for the latest contact information.

Safety Compliance

EC Declaration of Conformity– Low Voltage

The instrument meets intent of EC Directive 2014/35/EU for Product Safety. Compliance was demonstrated to the following specifications as listed in the Official Journal of the European Communities:

EN 61010-1:2010, Safety requirements for electrical equipment for measurement, control, and laboratory use – Part 1: General requirements

EN 61010-2:030:2010, Safety requirements for electrical equipment for measurement, control, and laboratory use – Part 2-030: Particular requirements for testing and measuring circuits

The design has been verified to conform to the following limits put forth by these standards:

- Mains Supply Connector: Overvoltage Category II, instrument intended to be supplied from the building wiring at utilization points (socket outlets and similar).
- Measuring Circuit Terminals: No rated measurement category. Terminals not intended to be directly connected to the mains supply.
- Pollution Degree 2, use in operating environment where normally only dry, non-conductive pollution occurs. Temporary conductivity caused by condensation should be expected.

U.S. Nationally Recognized Agency Certification

The instrument has been certified by Underwriters Laboratories (UL) to conform to the following safety standard and bears the UL Listing Mark:

UL 61010-1 Third Edition, Safety standard for electrical measuring and test equipment

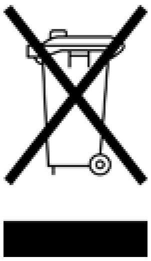
Canadian Certification

The instrument has been certified by Underwriters Laboratories (UL) to conform to the following safety standard and bears the cUL Listing Mark:

CAN/CSA-C22.2 No. 61010-1-12, Safety requirements for electrical equipment for measurement, control and laboratory use

Environmental Compliance

End-of-Life Handling



The instrument is marked with this symbol to indicate that it complies with the applicable European Union requirements to Directives 2012/19/EU and 2013/56/EU on Waste Electrical and Electronic Equipment (WEEE) and Batteries.

The product is subject to disposal and recycling regulations that vary by country and region. Many countries prohibit the disposal of waste electronic equipment in standard waste receptacles. For more information about proper disposal and recycling of your Teledyne LeCroy product, please visit teledynelecroy.com/recycle.

Restriction of Hazardous Substances (RoHS)

This instrument and its accessories conform to the 2011/65/EU RoHS2 Directive, inclusive of any further amendments or modifications of said Directive.

ISO Certification

Manufactured under an ISO 9000 Registered Quality Management System.

Intellectual Property

All patents pertaining to the HDO6000/HDO6000A can be found on our website at:

teledynelecroy.com/patents/

Index

TELEDYNE LECROY
Everywhereyoulook™

700 Chestnut Ridge Road
Chestnut Ridge, NY 10977
USA

teledynelecroy.com