



Operator's Manual

WaveSurfer 4000HD Oscilloscopes

Contents

Oscilloscope Overview and Set Up	1
Safety	1
Overview	4
Powering On/ Off	10
Software Activation	10
Language Selection	10
Connecting to Other Devices/ Systems	11
Configuring the Save/ User Button	12
Using MAUI	13
Touch Screen	13
MAUI with OneTouch	18
Working With Traces	22
Zooming	26
Acquisition	31
Auto Setup	31
Vertical	32
Digital (Mixed Signal)	35
Timebase	39
Trigger	45
Display	53
Display Set Up	53
Persistence Display	54
Cursors	57
Cursor Types	57
Apply and Position Cursors	59
Standard Cursors Dialog	60
XY Cursors Dialog	60
Measure	61
Measure Table	61
Parameter Set Up	62
List of Standard Measurements	64
Using Trends	67
Math	69
Math Function Set Up	69
Math Dialog	70
List of Standard Math Operators	71
Average Function	73
ERes Function	74
FFT Function	76
Rescale Function	78

Memory	83
Saving Memories	83
Restoring Memories	84
Analysis Tools	85
WaveScan	85
PASS/FAIL Testing	89
Saving Data (File Functions)	91
Save	91
Recall	97
LabNotebook	99
Share	101
Email Settings	102
Print	103
Using the File Browser	104
Utilities	107
Utilities Dialog	107
Disk Utilities	112
Preferences Dialogs	113
WaveSource Automatic Waveform Generator	117
Digital Voltmeter	119
Maintenance	121
Restore Default Setup	121
Firmware Update for CE Oscilloscopes	121
Technical Support	122
Returning a Product for Service	123
Certifications	125
EMC Compliance	125
Safety Compliance	126
Environmental Compliance	127
ISO Certification	127
Intellectual Property	127
Warranty	128
Index	129

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 instrument from inadvertent use.

Operating Environment

Temperature:	0 °C to 50 °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 $\leq$ 5 Vrms 1 M Ω coupling $\leq$ 400 Vpk max. (Peak AC $\leq$ 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 instrument using 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 ($\pm 5\%$) 100-120 VAC at 400 Hz ($\pm 5\%$)
Maximum Consumption:*	150 W (150 VA)
Nominal Consumption:	90 W (90 VA)
Standby Consumption:	4 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 against electric shock, connect to a mating outlet with a safety ground contact.

Overview

Front of Oscilloscope



- A. Touch screen display
- B. Front panel controls
- C. Standby Power button
- D. ProBus analog inputs (4)
- E. Mixed Signal interface
- F. USB 3.1 Gen 1 ports (2)
- G. Ground and Cal Out terminals
- H. Tilting feet

Back of Oscilloscope



A. Wave Source output

B. Kensington lock

C. Micro SD card (removable drive)

D. EXT trigger input

E. AUX output

F. HDMI out (to external monitor)

G. Ethernet (LAN) port

H. USBTMC port

I. USB 2.0 ports (2)

J. 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 front panel buttons light to indicate which functions and traces are active. The labels for Trigger, Horizontal, Vertical and Cursors/ Adjust will light in the color of the trace that is currently associated with these knobs. The Math, Zoom, and Mem indicators between the Horizontal and Vertical knobs will light when they are controlling a math function, zoom, or memory trace (instead of a channel).

Miscellaneous Controls

Auto Setup performs an [Auto Setup](#). After the first press, you will be prompted for a confirmation. Press the button again or use the touch screen to confirm.

Default Setup resets the oscilloscope to the factory default configuration. After the first press, you will be prompted for a confirmation. Press the button again or use the touch screen to confirm.

Touch Screen toggles on/off touch screen functionality.

Save/ User can be configured as you choose, for example: save a screen capture using your current [Screen Image Preferences](#); save other file types; or clear sweeps and restart measurement counters. See [Configuring the Save/ User Button](#) for instructions on making the selection.

Trigger Controls

The front panel **Level knob** changes the trigger threshold level (V or A). The number is shown on the Trigger descriptor box. Pushing the knob sets the trigger level to the 50% amplitude of the input signal.

The **READY** indicator lights when the trigger is armed. The **TRIGD** indicator lights when a trigger occurs.

Setup opens the Trigger Setup dialog. Press it again to close the dialog.

Auto starts Auto trigger mode, which triggers the oscilloscope after a set time, even if the trigger conditions are not met.

Normal starts Normal trigger mode, which triggers the oscilloscope each time the signal on the triggering channel meets the trigger conditions.

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

Stop prevents the scope from triggering on a signal. If you boot up the instrument with the trigger in **Stop** mode, a "No trace available" message is shown. Select another mode to begin acquisition.

Horizontal Controls

Turn the **Delay knob** to raise/lower the Trigger Delay value (S). Push the knob to reset Delay to zero.

If the trace source is an input channel, turn the **Horizontal Adjust knob** to set the Time/division (S) of the oscilloscope acquisition system. The value is shown on the Timebase descriptor box. When using this control, the oscilloscope allocates memory as needed to maintain the highest sample rate possible for the timebase setting. If the active trace is a math, zoom or memory, use this knob to change the Horizontal Scale, effectively "zooming" in or out. The value is shown on the corresponding descriptor box.

Push the knob to adjust scale in fine increments; push it again to return to stepped increments.

Vertical Controls

The **Offset knob** adjusts the zero level of the trace (this makes it appear to move up or down relative to the center axis of the grid). The value appears on the trace descriptor box. Push it to reset Offset to zero.

The **Vertical Adjust knob** sets Volts/div for channel traces or Vertical Scale for math, zoom and memory traces. The value appears on the trace descriptor box. Push the knob to toggle between fine and stepped increments.

The **Channel buttons** (1-4) 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.

The **Zoom (Z) button** creates a Quick Zoom of each open trace. The resulting zoom traces are 1/10 of the source channel scale and centered on the display. Use the Horizontal knobs to adjust this, or touch the zoom descriptor box twice to display the zoom dialog controls. Touch the Zoom button again to turn off the zooms.

The **Digital (D) button** enables digital input on oscilloscopes with the MSO option.

Cursor Controls

Cursors identify specific vertical and horizontal values on the waveform. The white cursor markers help make these points more visible, as well as provide a simple way to reposition them. A readout of the vertical values appears on the trace descriptor box, while the horizontal values appear below the Timebase 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.

Press the **Cursor Type** button to apply or remove cursors. Continue pressing to cycle through all cursor types until the desired type is found ("Off" occurs when no cursors are visible).

Turn the **Coarse** knob to adjust the position of absolute cursors (dashed-dotted lines). Push it to return the cursor to the default 2.5 division setting. If you're using the Both Rel cursor type, each push will select the other absolute cursor. With the correct cursor selected (highlighted), turn the knob to adjust the position.

Turn the **Fine** knob to adjust the position of relative cursors (dashed lines). Push it to return the cursor to the default 7.5 division setting, or to select the other relative cursor when using the Both Rel cursor type.

When Cursors are off and a data entry field is selected, these knobs serve as the **Adjust knobs**. Turn the left knob to make coarse (1-, 2-, 5-, 10-stepped) adjustments; turn the right knob to make fine (single-increment) adjustments. Push either knob to return to the default setting.

Signal Interfaces

Teledyne LeCroy instruments offer a variety of interfaces to input analog or digital signals.

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 M Ω 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 (Cn) 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 with 1 M Ω 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.

This input has 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 MSO option 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

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.

Micro SD Card

The Micro SD Card acts as the oscilloscope's removable hard drive. Use it to store and easily share setup files, waveform files, LabNotebooks, and other user data.

To remove the card, push in and release. The card should partially pop out, at which point it can be pulled out fully.

To replace the card, push it into the slot until you hear it click.



Note: When using the oscilloscope Disk Utilities, the Micro SD card is labeled Storage Card, while a connected USB drive is labeled USB Disk.

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 power on or calibrate with a signal attached.

Software Activation

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

Free firmware updates are available periodically from the Teledyne LeCroy website at:

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.

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.

Connecting to Other Devices/ Systems

After start up, configure the oscilloscope connections using the menu options listed below.

LAN

The oscilloscope is preset to accept DHCP network addressing over a TCP/IP connection. Connect an Ethernet cable from the port on the back panel to a network access device.

Go to **Utilities > Utilities Setup > Remote** to find the IP Address.

To configure a Static IP address:

1. Touch **Net Connections** on the Remote dialog.
2. Touch the **Mini-port icon** (right) to open the driver settings.
3. Select **Specify an IP address** and enter the new address.

To configure [Email](#), go to **File > File Sharing > Email Settings**.

Remote Control

You can remote control the instrument using TCP/IP over LAN. Be sure the instrument is on the same subnet as the controller.



Note: You can also use the LAN port to make a direct connection, but depending on the controller, you may have to use a cross-over cable.

The WaveSurfer also supports remote control via USBTMC or GPIB (with the optional USB-GPIB adapter). To change the remote control setting from the default TCP/IP, go to **Utilities > Utilities Setup > Remote**.

USB Peripherals

Connect peripherals (e.g., mouse, keyboard) to any **USB host port** on the front or back of the instrument. These connections are "plug-and-play" and do not require further configuration.

External Monitor

WaveSurfer 4000HD supports external monitors with 1280x800 ppi resolution. Connect the monitor cable to the **HDMI video output** on the back of the instrument. The connection is "plug-and-play" and does not require any further configuration.

Printer

WaveSurfer 4000HD supports PictBridge-compliant printers. Connect the printer to any host USB port. Go to **File > Print Setup** to configure printer settings.

Trigger Out

To send a trigger out pulse to another device, connect a BNC cable from **Aux Out** on the back of the instrument to the other device.

WaveSource Waveform Generator

Connect a BNC cable from the WaveSource Output on the back of the instrument to the signal input device. Go to **Utilities > WaveSource** to configure the signal.

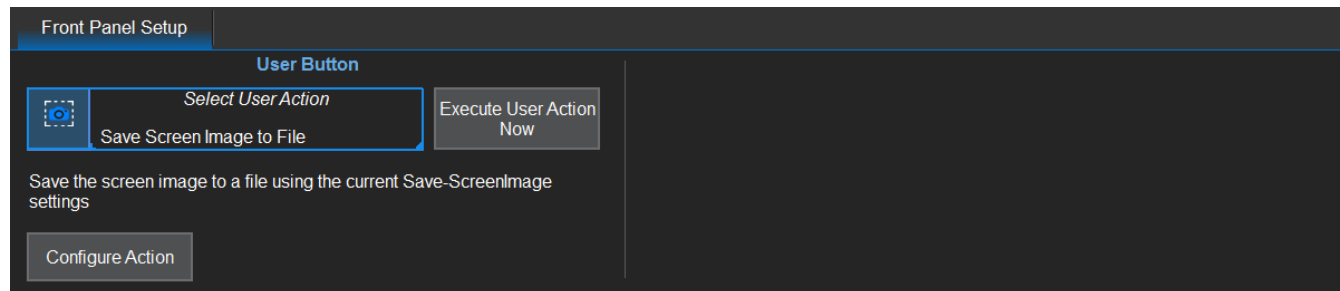


Note: WaveSource is optional and requires an activated license key.

Configuring the Save/ User Button

The front panel Save/ User button can be configured to perform your choice of actions, such as saving a screen image or other file type.

1. Go to **Utilities > Front Panel Setup**, then touch **Select User Action** and make your User button selection.
2. Optionally, touch **Configure Action** to make any other setting related to that action, such as Screen Image or Print preferences.



Using MAUI

MAUI (Most Advanced User Interface) is Teledyne LeCroy's unique oscilloscope user interface.

Touch Screen

The oscilloscope features a capacitive touch screen that supports fluid, tablet-like response to gestures.



Note: Use your finger or a capacitive stylus (not included) to interact with the touch screen. A regular stylus will not work.

The entire display area is active. 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](#)

[Grid](#)

[Descriptor boxes](#)

[Dialogs](#)

[Message bar](#)

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.

The **Clear Sweeps** button to the right of the menu bar clears acquired waveforms from the history buffer and resets all measurement counters and histograms.

Grids

The grids display the waveform traces. Every grid is 8 vertical divisions representing the full number of vertical levels and 10 horizontal divisions. The value represented by each division depends on the Vertical and Horizontal Scale of the traces that appear on the grid.

The grid region can be divided up to three times to show channel (Cn), math (Fn), and zoom (Zn) traces on different grids. In Auto Grid mode, it will divide automatically as needed when new types of traces are turned on. Two additional grid styles allow you to display XY traces, as well as voltage-time traces on separate grids. 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.

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.

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 time and unit represented by a grid division. They update dynamically as you pan the trace or change the scale. Originally shown in absolute values, the labels show delta from 0 (center) when the number of significant digits grows too large. To remove them, 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 Time indicator to a time not shown on the grid. All Delay values are shown on the Timebase Descriptor Box.



Trigger Level at the right edge of the grid tracks the last triggered voltage level. If you change the level, a hollow triangle of the same color appears at the new level until it has triggered. 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.

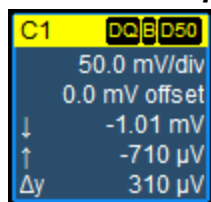
Descriptor Boxes

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, twice to open the setup dialog.
- **Configure**—drag-and-drop descriptor boxes to change source or copy setups (with OneTouch).

Besides trace descriptor boxes, there are also Timebase and Trigger descriptor boxes summarizing the acquisition settings shared by all channels, which also open the corresponding setup dialogs.

Channel 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 (in Sequence mode).

Codes are used to indicate pre-processing and coupling that has been applied to the input. The short form is used when several processes are in effect.

Symbols on Descriptor Boxes

Pre-Processing Type	Long Form	Short Form
Deskew	DSQ	DQ
Bandwidth Limiting	BWL	B
Coupling	DC50, DC1M, AC1M or GND	D50, D1, A1 or G

Other Trace Descriptor Boxes

F1 trend(P3)	Z1 zoom(C1)	M1
50.0 ps/div	200 mV/div	200 mV/div
20.0 #/div	5.00 ns/div	50.0 ns/div
0 S		

Similar descriptor boxes appear for math (F_n), zoom (Z_n), and memory (M_n) traces. These descriptor boxes show any Horizontal scaling that differs from the signal timebase. Units will be automatically adjusted for the type

of trace.

Timebase and Trigger Descriptor Boxes

HD	Timebase	0 ns	Trigger	C1 DC
12 Bits	50.0 ns/div	Stop	0.0 mV	
	2.5 kS	5 GS/s	Edge	Positive
X1 = -125.0 ns ΔX = 250.0 ns				
X2 = 125.0 ns 1/ΔX = 4.000 MHz				

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).

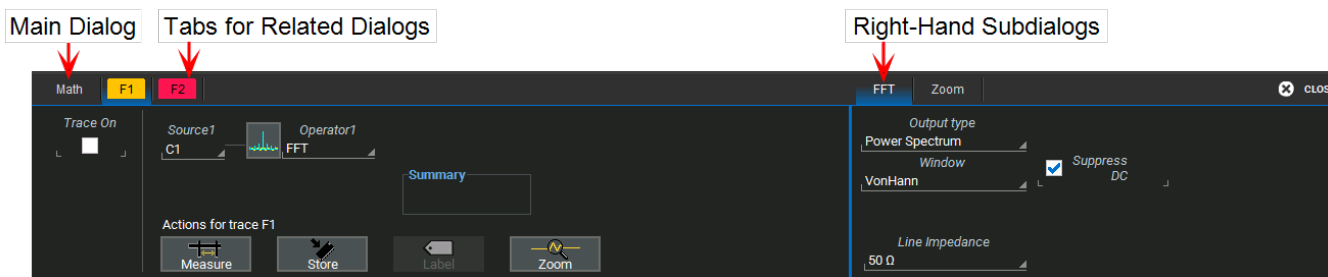
Trigger descriptor box shows: (clockwise from top right)

Trigger Source and Coupling, Trigger Level (V), Slope/ Polarity, Trigger Type, Trigger 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.

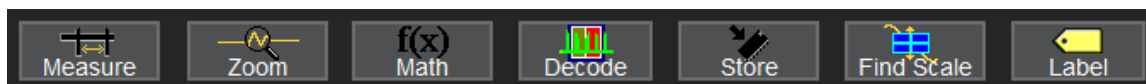


Right-Hand Subdialogs

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.

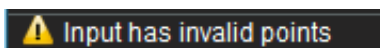
Store loads the active trace into the corresponding memory location (C1, F1 and Z1 to M1; C2, F2 and Z2 to M2, etc.).

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

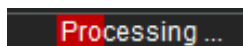
Label opens the Label pop-up to annotate the active trace.

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 and swipe 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.

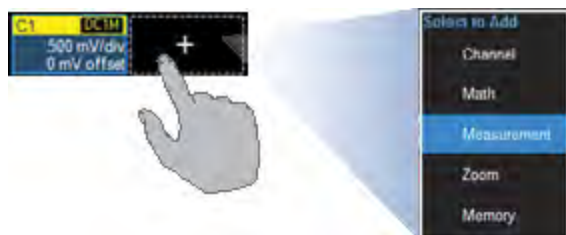
Turn On

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.



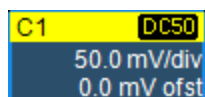
If there is no descriptor box of the desired type on the screen to drag, touch the Add New box and choose the trace type from the pop-up menu.

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.

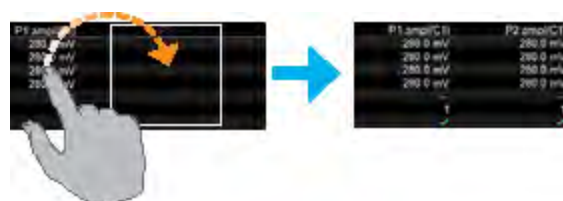


Copy Setups

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 (Pn)**, drag-and-drop the source column onto the target column of the Measure 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.

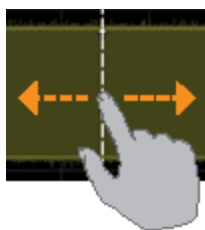


To **change the source of a measurement**, drag-and-drop the descriptor box of the desired source onto the parameter (Pn) column of the Measure table.

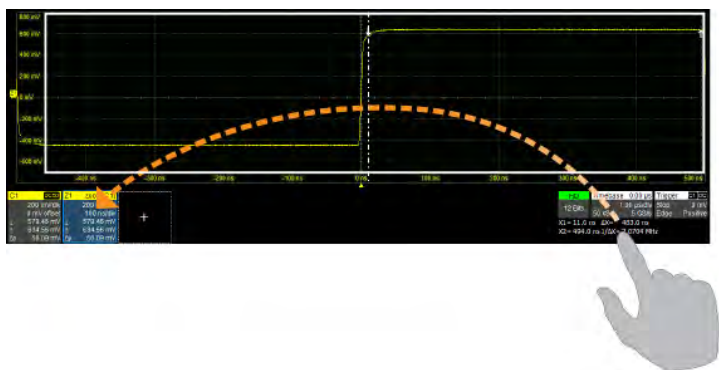


Position Cursors

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

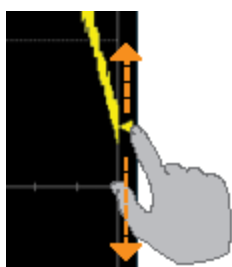


To **place horizontal cursors** on zooms or other calculated traces where the source Horizontal Scale has forced cursors off the grid, drag the cursor readout from below the Timebase descriptor to the grid where you wish to place the cursors. The cursors are set at the 2.5 and 7.5 divisions of the grid.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

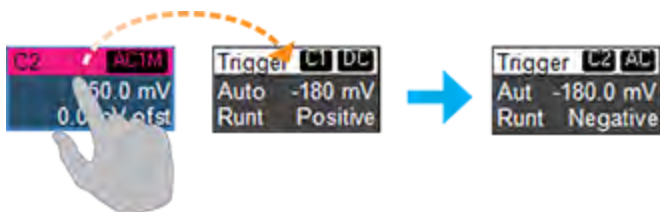


Change Trigger

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 (Cn) 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 (Mn) descriptor box.



Scroll

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



Pan 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.



Working With Traces

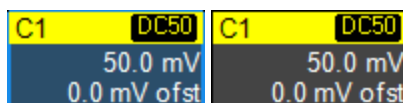
Traces are the visible representations of waveforms that appear on the display grid. They may show live inputs (Cn , Digital n), a math function applied to a waveform (Fn), a stored memory of a waveform (Mn), a zoom of a waveform (Zn), or the processing results of special analysis software.

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.

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.

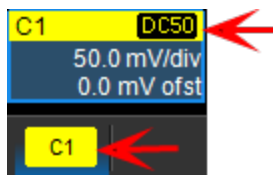
Active Trace

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 (Cn) 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 > Digitaln Setup** then check **Group** on the Digitaln dialog.

Turn off the trace, clear the Group checkbox.

Turn On/ Off Zoom Trace

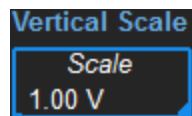
See [Creating Zooms](#).

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

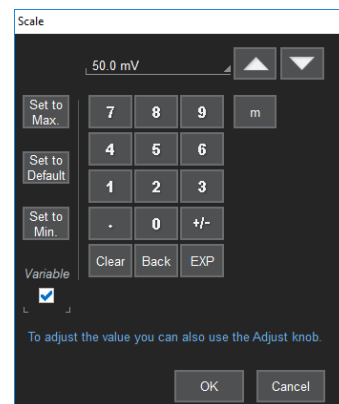


To adjust Vertical Scale and Offset, or Horizontal Scale and Delay, just activate the trace and use the front panel knobs. To make other adjustments—such as units—touch the trace descriptor box twice to open the appropriate setup dialog.

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.



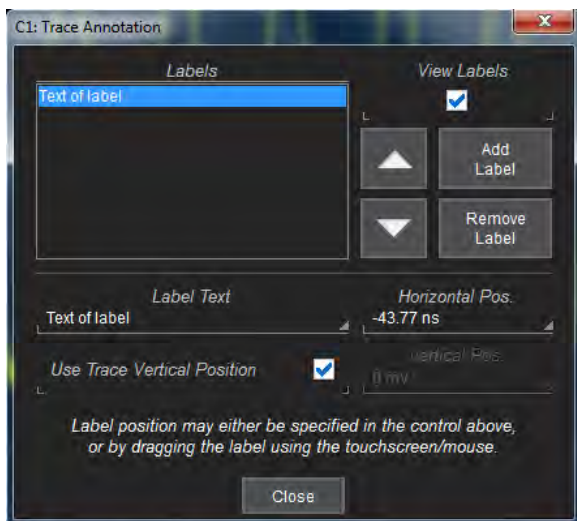
Labeling Traces



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.

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](#). All enabled zooms open in the same grid at the same horizontal scale.

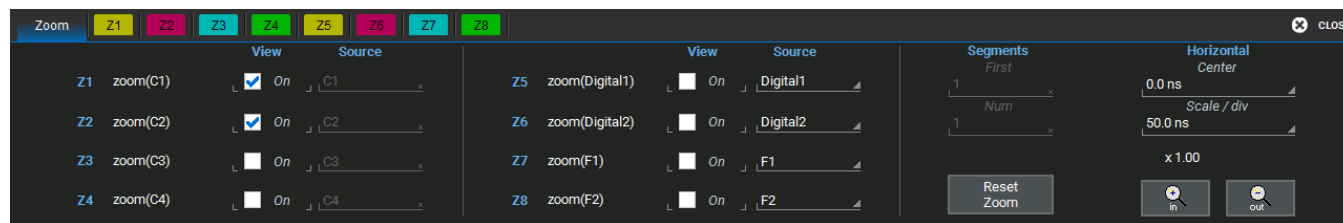
The current settings for each zoom trace can be seen on the [Zn dialogs](#).

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 [Fn dialogs](#)
- Memory traces on [Mn dialogs](#)

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

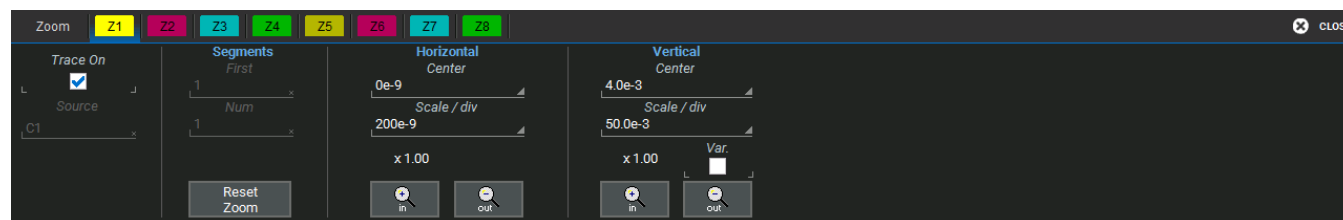


The main Zoom dialog contains selection boxes for turning on/off and changing the source of a zoom. There are also options to:

- **Reset Zoom**, return all zooms to x1 magnification.
- Simultaneously change **Center** and **Horizontal Scale** for all active zooms, either by setting the values on the dialog or touching the **In** and **Out** buttons.

Zn Dialog

Each Zn 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.



Note: Z1 through Z4 are permanently assigned to C1 through C4 respectively.

Segment Controls

These controls are used only in [Sequence Sampling Mode](#).

Zoom Factor Controls

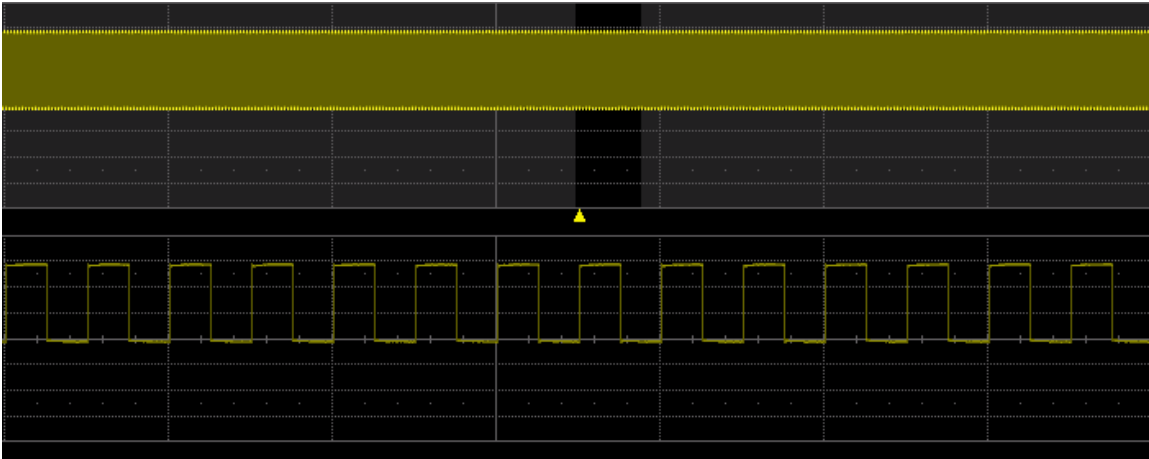
- **Out** and **In** buttons increase/decrease zoom magnification and consequently change the Horizontal and Vertical Scale settings. Touch either button until you've achieved the desired level.
- **Var.checkbox** enables zooming in single increments.
- **Horizontal Scale/ div** sets the time represented by each horizontal division of the grid. It is the equivalent of Time/ div in channel traces.
- **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. The horizontal center is the same for all channel zoom traces.
- **Reset Zoom** returns the zoom to x1 magnification.



Tip: On WaveSurfer oscilloscopes, all channel zooms (Z1-Z4) are displayed in the same grid at the same horizontal scale. When zooming digital, math or memory traces, the horizontal and vertical scale may be set independently for each zoom.

Creating Zooms

Any type of trace can be zoomed by creating a new zoom trace (Z_n) following the procedures here. All zoom traces open in the same grid, with the zoomed portion of the source trace left unshaded.



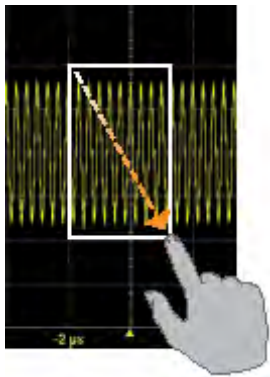
Zoomed area of source trace left unshaded.

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. Each channel is assigned to the equivalent numbered zoom trace (C1 to Z1, etc.).

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

Manually Create Zoom



To manually create a zoom of a channel trace, touch-and-drag diagonally to draw a rectangle around any part of the source trace. The horizontal area within the rectangle is expanded, while the vertical area is rescaled proportionally. The degree of vertical and horizontal magnification, therefore, depends on the size of the rectangle that you draw.

Doing this over a zoom, math or memory trace rescales the same trace, rather than creating a new zoom. To zoom these types of traces, go to **Math > Zoom Setup** and choose it as the Source of Z5 through Z8, or use the **Zoom action button** at the bottom of the trace setup dialog.

You can also create new zooms by creating 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 *Zn* slot, ready to be turned on again when desired.

To close the zoom, clear the **Trace On** box on the Zoom or *Zn* dialog.

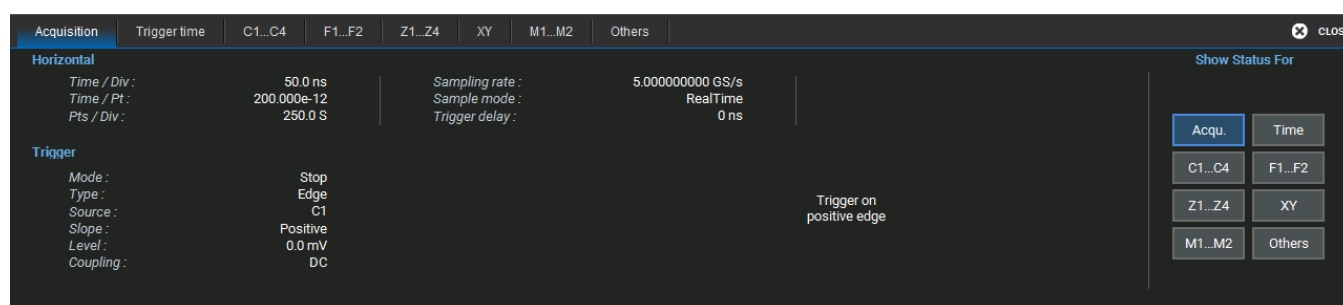
Acquisition

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.

Vertical

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



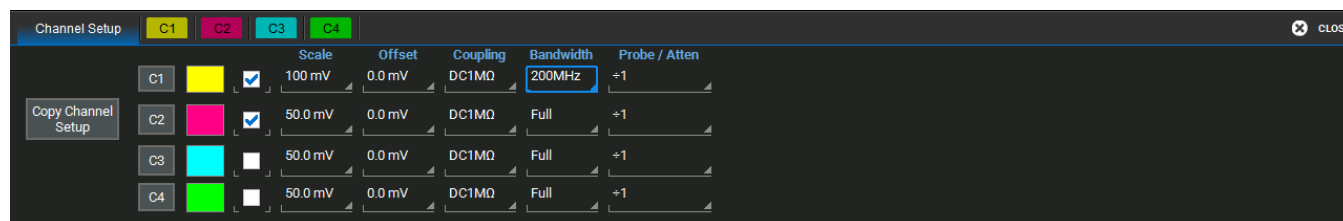
Note: While Digital settings can be accessed through the Vertical menu on Mixed Signal oscilloscopes, they are handled quite differently. See [Digital](#).

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 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**.

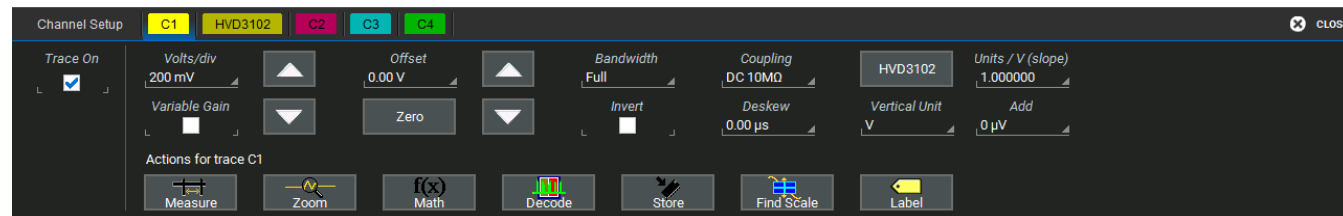


Tip: On instruments with OneTouch, you can copy settings from one channel to another just by dragging the source channel descriptor box onto the target channel descriptor box.

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.



Vertical Settings

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

Volts/div 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 (or 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.

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.

Probe Attenuation and Deskew

Probe **Attenuation** values for third-party probes may be entered manually on the *Cn* 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 *Cn* tab. Enter Attenuation on the Probe dialog.

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.

Rescale

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



Tip: These settings do the same as the [Rescale](#) math function ($y=ax+b$, where the original value is x , Units/V is a , and Add is b), only applied directly to the channel trace.

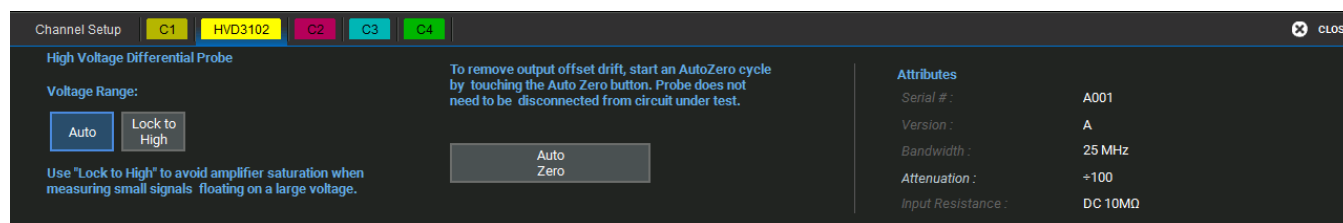
Vertical Units may be changed from Volts (V) to Amperes (A).

Probe Dialog

The Probe Dialog immediately to the right of the *Cn* 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.

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:

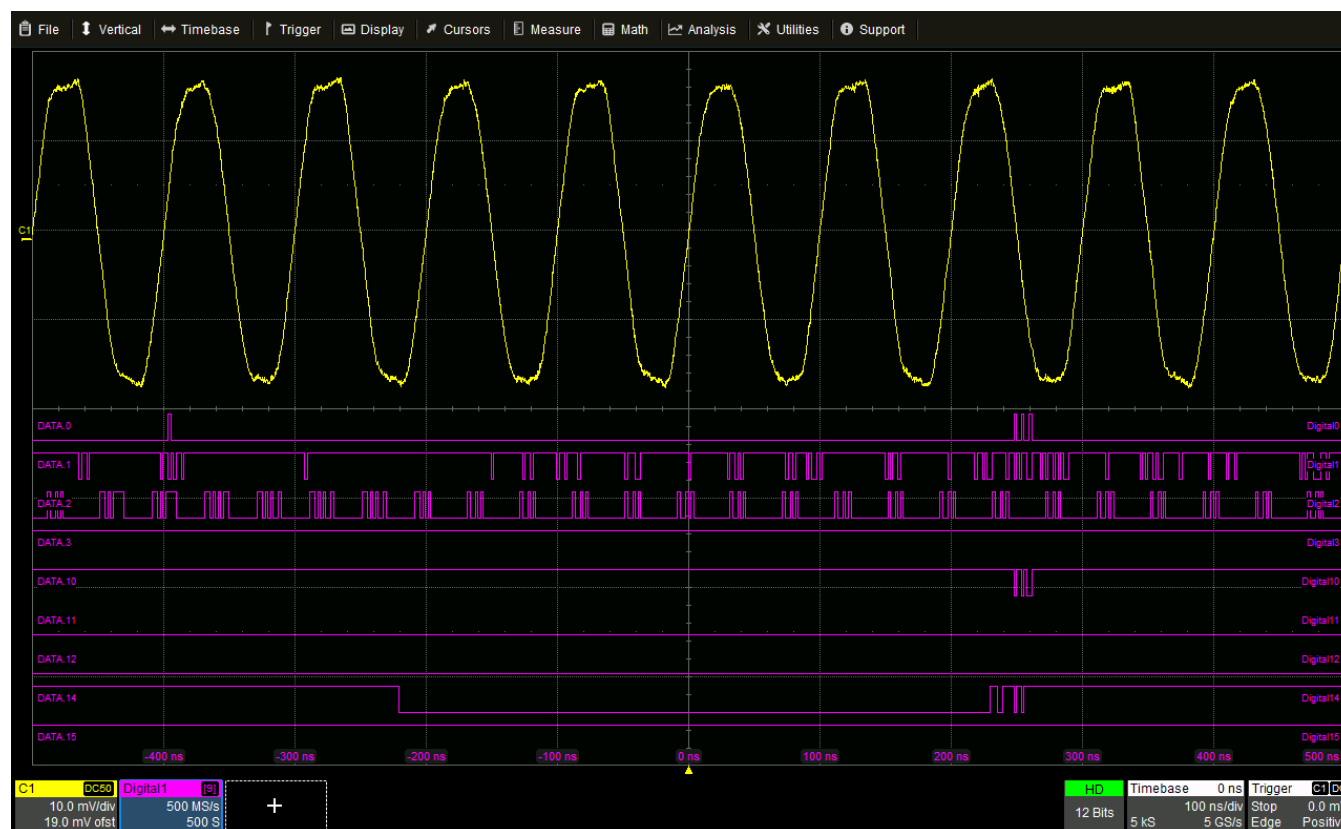
- **Auto** automatically raises attenuation when V/div is >7.9 or lowers attenuation when V/div is <7.9 , allowing you to properly view the input waveform.
- **Lock to High** locks attenuation to the highest setting, regardless of the V/div setting. Maintaining a high attenuation will allow small signals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

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.



Nine digital lines displayed with the Vertical Position 0 (center), sharing the grid with an analog trace. 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.



High



Low



Transitioning

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 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

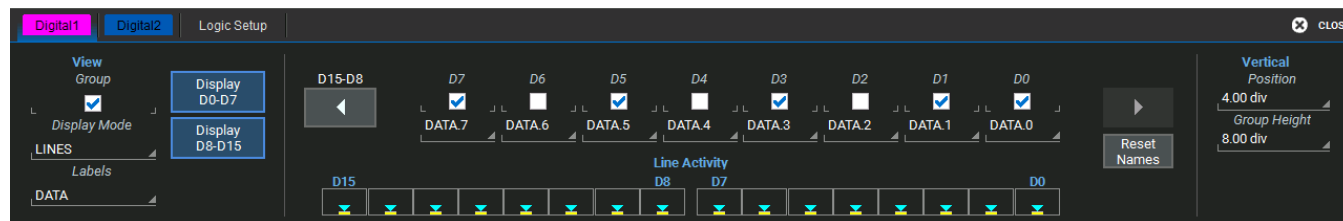
To connect the leadset to the instrument, push the connector into the Mixed Signal interface below the front panel until you hear a click.

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:

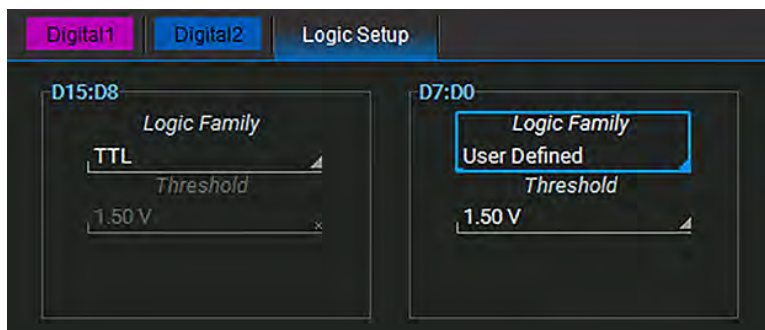
1. From the menu bar, choose **Vertical > Digital n Setup**.
2. On the Digital n set up dialog, check the boxes for all the lines that comprise the group. Touch the Right and Left Arrow buttons to switch between digital banks as you make line selections.



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

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

- When you're finished on the *DigitalIn* dialog, open **Logic Setup** and choose the **Logic Family** that applies to each digital bank, or set custom **Threshold** values.



Digital Display Setup

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

- 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.
- 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.
- 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 *DigitalIn* dialog.

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.

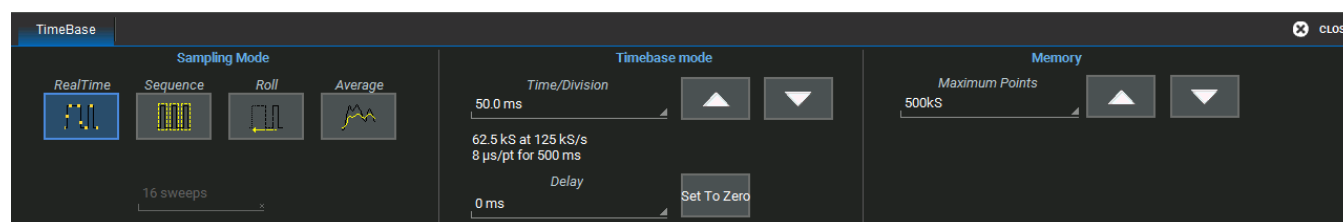
Timebase

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.

Memory

Maximum Points is the maximum number of samples taken per acquisition. The actual number of samples acquired can be lower due to the other timebase settings.

The oscilloscope will allot the maximum memory and sample rate possible based on the activity within each *pair* of channels. As long as only one channel in each of the C1-C2 and C3-C4 pairs is turned on, the interleaved rate is used. Turning on both channels in either pair reduces memory and sample rate to the standard, 4-channel specification.

Sampling Modes

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

Average Sampling Mode

Average sampling mode calculates the average value for each captured point over a specified number of acquisitions. Each individual acquisition uses Real Time mode and the results are averaged together. Average mode can be used to reduce random noise in repeating signals.

When selecting Average sampling mode, also select the number of Sweeps to calculate in the Average.

The Max Memory Length you can set for Average sampling mode is 10 kpts. This limit applies only to the hardware acquisition system. You can apply the Average math function to larger acquisitions.

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:

$$\begin{aligned}\text{Capture Interval} &= 1/(\text{Sample Rate} \times \text{Memory}) \\ \text{Capture Interval}/10 &= \text{Time Per Division}\end{aligned}$$

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.

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 50 ms/div or more 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.

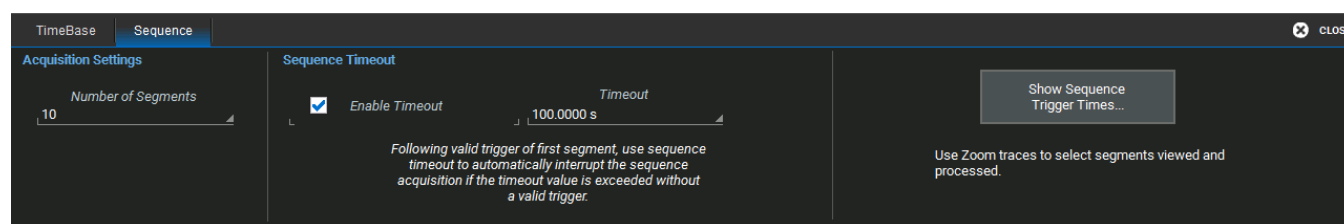
Sequence Sampling Mode

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 are arranged adjacent to one another, forming the waveform display of a typical acquisition.

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. 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 **Zn** dialog. Touch the **Zn** descriptor box to display the dialog.



Tip: By setting the Num to 1, you can use the front panel Fine 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 total number of segments displayed. As with all other zoom traces, the zoomed segments are highlighted on the source trace.

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**.

History Mode

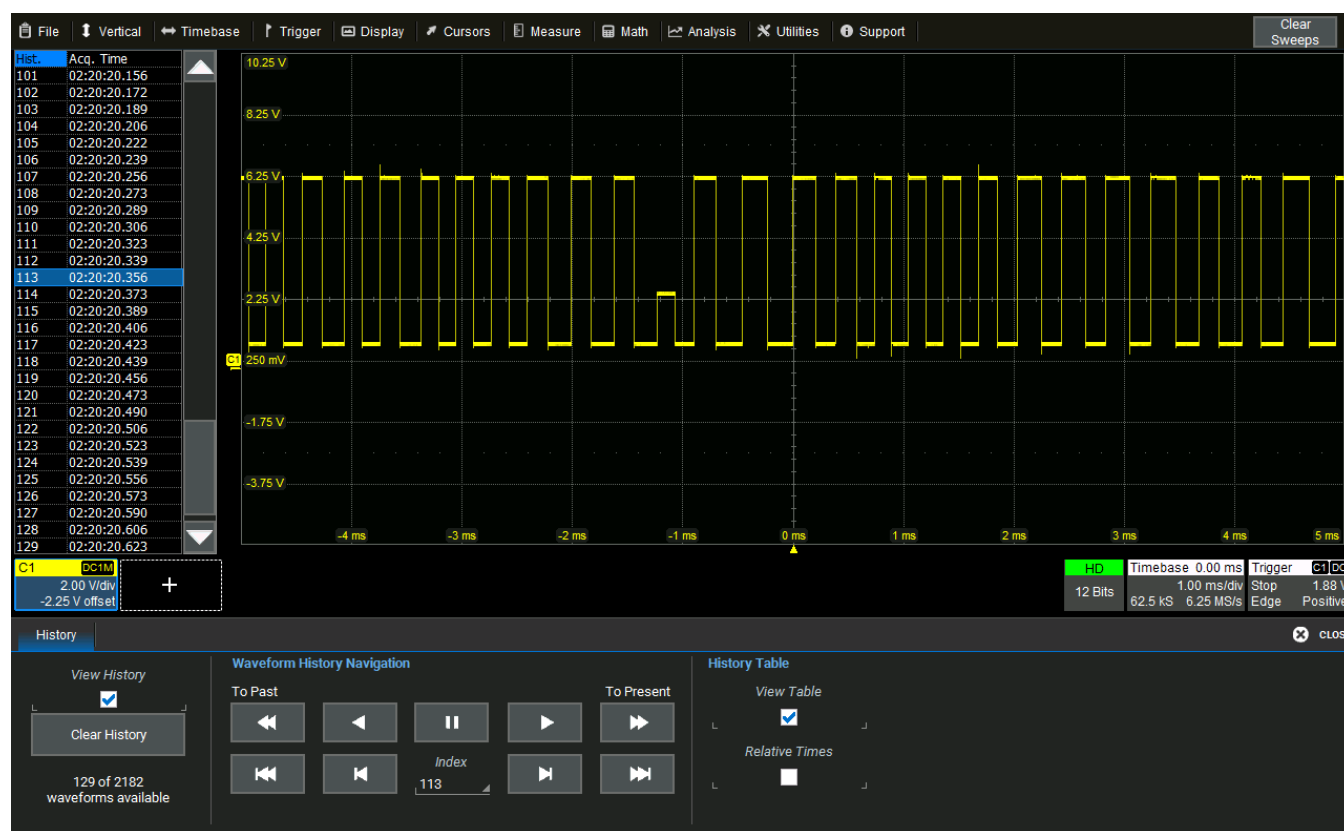
History Mode allows you to review any acquisition saved in the history buffer, which automatically stores all acquisition records until full. Not only can individual acquisitions be restored to the grid, you can "scroll" backward and forward through the history at varying speeds to capture changes in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

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, restart acquisition by pressing one of the front panel Trigger Mode buttons.



Note: History Mode does not work with Average or Sequence Mode acquisitions, Interpolation set on the input channel, or any type of channel interleaving.



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 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 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.

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.

Qualified

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.

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.

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.

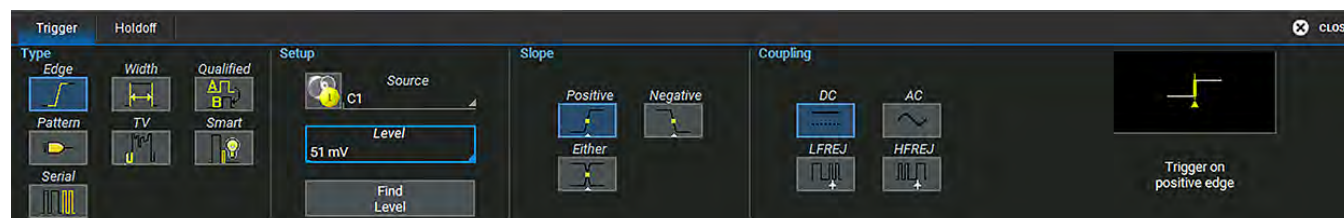
Slew Rate triggers when the rising or falling edge of a pulse crosses an upper and a lower level. The pulse edge must cross the thresholds faster or slower than a selected period of time.

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).

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**. Usually, there will be a separate Levels tab for these settings.

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). Choosing a Slope of Either will cause the trigger to fire when the Level is reached on either edge of the waveform.

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).

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.

1. On the Trigger dialog, select **Pattern** trigger type. Open the **Digital Pattern** dialog.



2. Enter the hexadecimal value of the pattern in **Hex**. Lines will take a logical 1, 0, or X ("Don't Care") according to the pattern. Disabled lines will remain X.

OR

Touch the **Dn button** for each active line, 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 Don't Care (X) selected for any line you wish to exclude from the pattern. Use the Left and Right Arrow buttons to display lines in other digital banks.

3. To add analog channels to the pattern, touch the Left Arrow button until you see buttons for C1-C4. Touch the buttons and choose High, Low, or Don't Care to set the pattern.



Note: Analog patterns always assume a logical "And" when combined with any digital pattern. Both conditions must be true for the trigger to fire.

4. To set a **Time Condition** in which the pattern must occur once the trigger is armed, choose the operator:
 - **Less Than** to trigger only if the pattern occurs before the time set.
 - **Greater Than** to trigger only if the pattern occurs after the time set.
 - **In Range** to set a time window in which the pattern must occur.
 - **Out Range** to set a time window outside which the pattern must occur.
5. Open the **Levels dialog** and select a **Logic Family** for each digital bank from which you've selected lines. To set a custom threshold, choose Logic Family User Defined, then enter the **Threshold** voltage.
6. For analog channels, enter the threshold voltage for each channel in the pattern separately.

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.

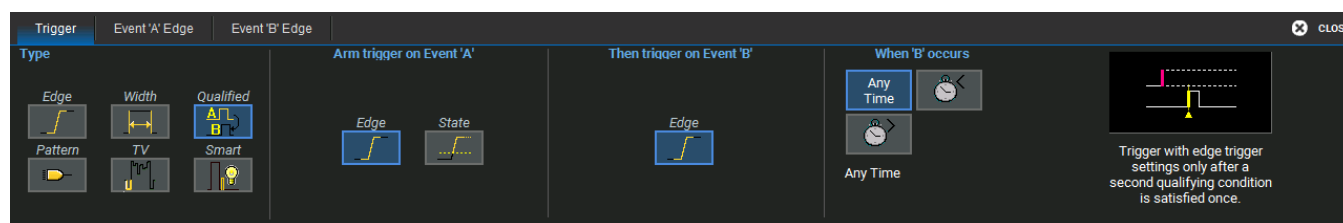
Serial Trigger

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.

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 Edgetrigger 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 **Qualified** trigger type to display the controls.



Besides an **Edge** level, the arming event (A) may be a **State**, any voltage measured above or below a threshold Level.

A time window may be added to the trigger conditions by using the **When B Occurs** buttons:

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

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.

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.

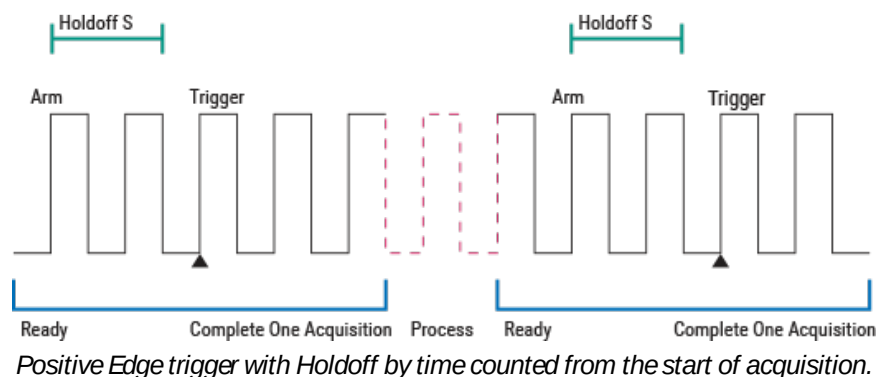


Note: [Qualified](#) triggers operate using time or event conditions similar to Holdoff, but arm and trigger differently.

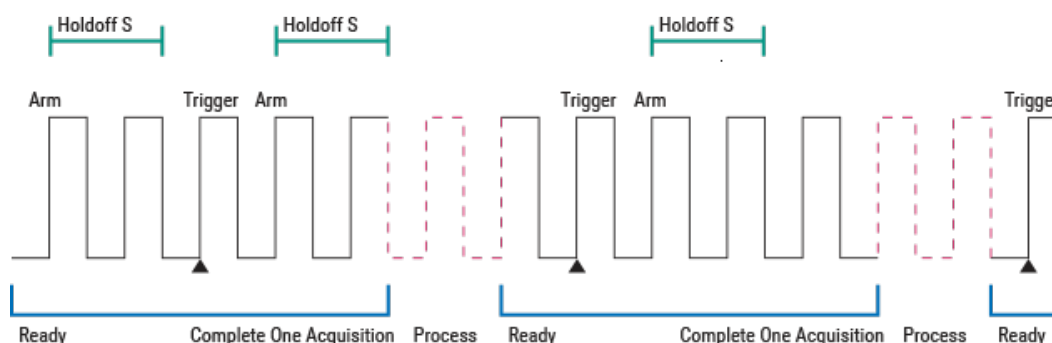
Hold Off by Time

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.



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.



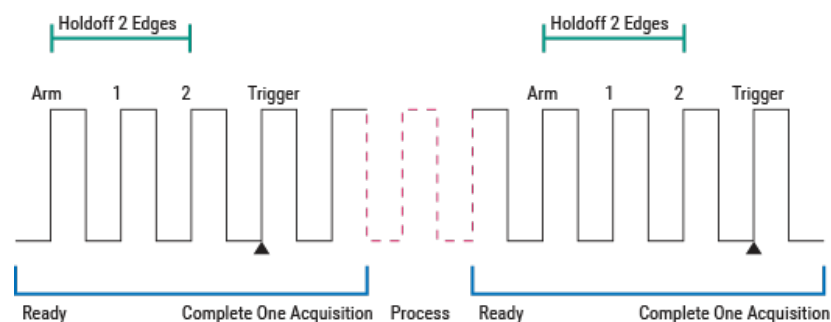
Note: Because there is only one trigger per acquisition, the trigger event will always belongs to the new acquisition. The processing time shown here is for purposes of illustration only.

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

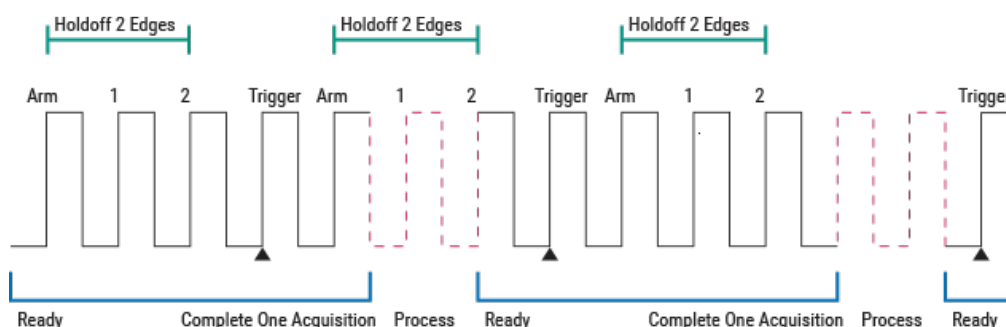
Events refers to the number of times the trigger conditions have been met following the arming event.

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.

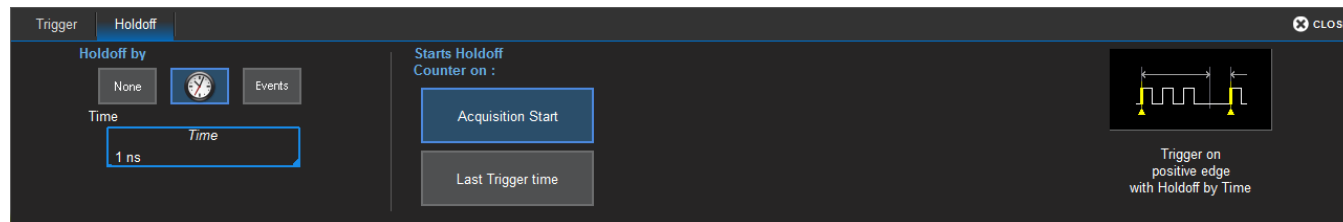
As with Holdoff by time, when a Holdoff by events is counted from the last trigger time, the oscilloscope re-arms immediately following the trigger and begins the Holdoff count. If the count is satisfied by the time the oscilloscope is again ready, the trigger occurs on the next event at the start of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

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.

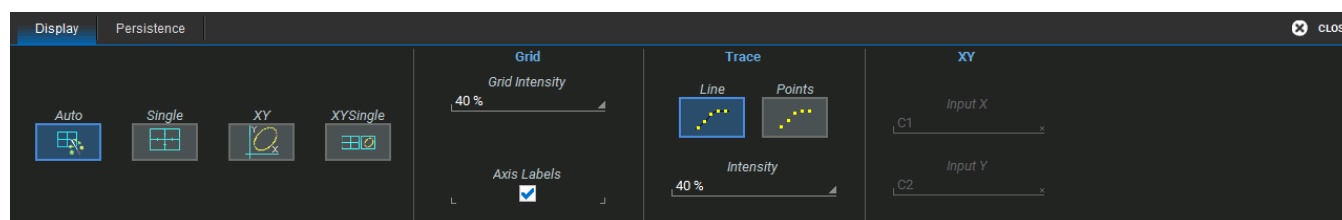
Auto Grid is enabled by default. This feature divides the screen as needed when new traces open. WaveSurfer oscilloscopes may be divided into a maximum of three grids—one each for channels/ memories, math functions, and zooms—that each represent the full number of vertical levels. All traces of the same type appear on the same grid.

Two special grid layouts are available: XY Grid, which puts the oscilloscope in XY mode, and XY Single Grid, which creates one XY grid and one single grid for the rest of your traces.

To display all types of traces on a single grid, choose **Single Grid** from the Display dialog.

Display Set Up

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



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.

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
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.

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 $\leq 40\%$ 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.

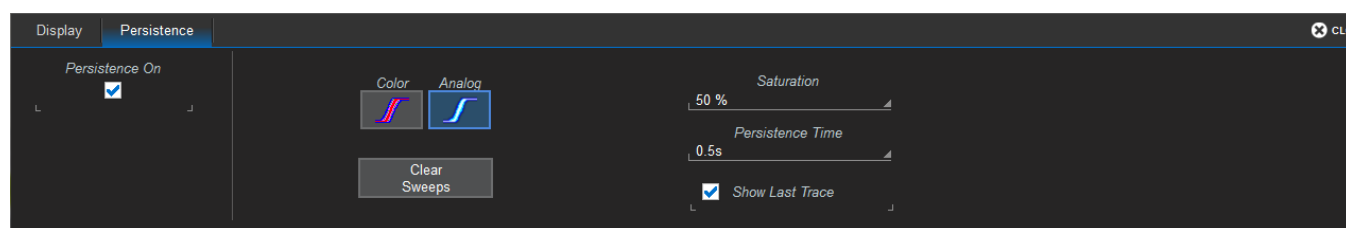
Choose an XY grid mode and select the sources for **Input X** and **Input Y**.

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.

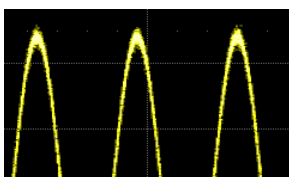
The different persistence modes show the most frequent signal path in three-dimensional intensities of the same color (Analog), or in a graded spectrum of colors (Color).

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

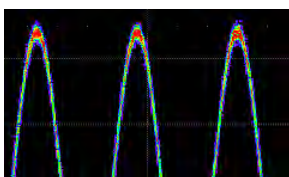


Apply Persistence

1. Check **Persistence On**.
2. 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.

3. 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.
4. In **Persistence Time**, enter the duration of time (in seconds) after which persistence data is erased from the display.
5. Choose to superimpose the last waveform over the persistence display by selecting **Show Last Trace**.

Remove Persistence

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

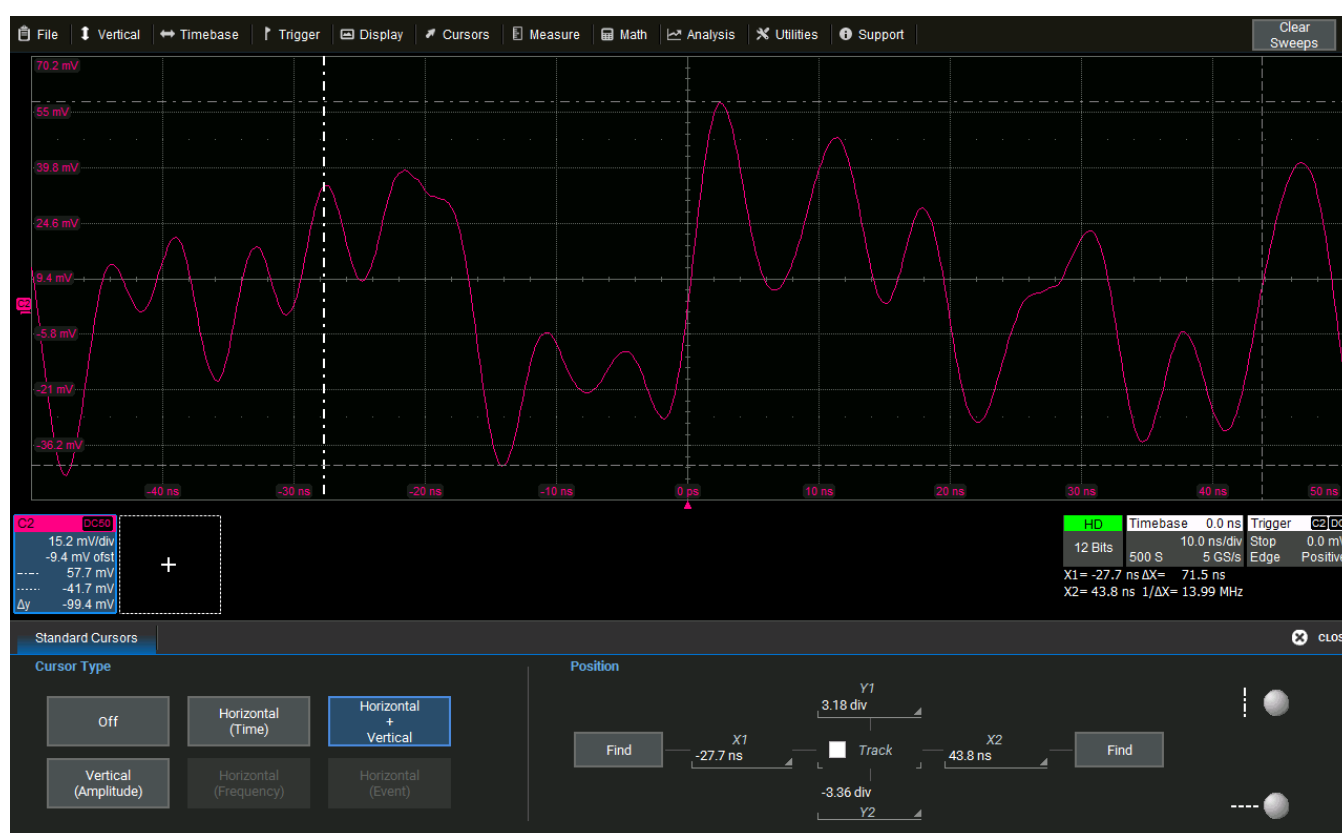
Cursors

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 Cursors.

The **Horizontal (Time)** 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 time where each cursor intersects the x-axis (X1 and X2)
- The difference of $X2 - X1$ (ΔX)
- The frequency in Hz calculated from the delta time ($1/(\Delta X)$).

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 shows the difference in vertical value where each cursor intersects the source trace (shown 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.

Two other Horizontal cursors are offered only in cases where the x-axis represents units other than time:

The **Horizontal (Frequency)** cursor works the same as the Horizontal (Time) cursor, except that it is placed on waveforms that have frequency (Hz) on the x-axis, such as FFTs.

The **Horizontal (Event)** cursor also works the same as the Horizontal (Time) cursor, but is placed only on Trend waveforms, where the x-axis represents the number of the measurement event.

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.

The **Vertical (Amplitude)** cursor displays two lines: the dashed-dotted line is $Y1$, and the dashed line is $Y2$. The readout on the source trace descriptor box shows the vertical values where $Y1$ and $Y2$ intersect the y-axis, and the difference of $Y1 - Y2$ (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.

Combination Cursors

The **Horizontal + Vertical** option places both Vertical (Amplitude) and Horizontal (Time) cursors together. The vertical readout on the source trace descriptor will be the same as for the Vertical (Amplitude) cursor, while the horizontal readout below the Timebase descriptor will be the same as for the Horizontal (Time) 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.

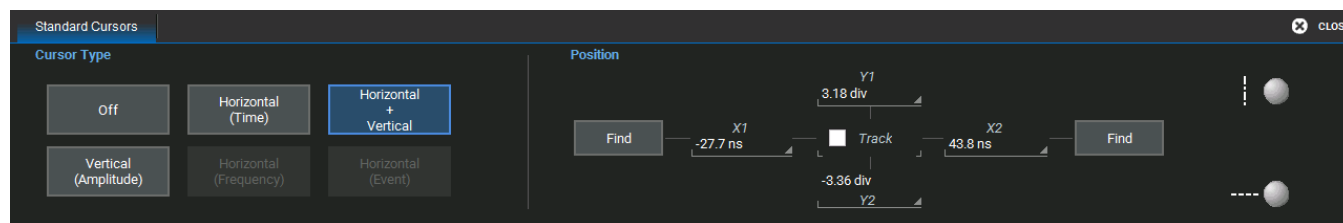
When there are multiple traces on the same grid, first bring the desired trace to the foreground by touching the trace or its descriptor box. The Cursor knob will only operate on the foreground trace.

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 2.5 and 7.5 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 front 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. **Off** disables the cursor display.

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.

- **X1** and **X2** sets the position of Horizontal cursors. They may be entered as time or a fraction of a division.
- **Y1** and **Y2** sets the position of Vertical cursors, entered as a fraction of a division.

Track locks 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.

Find places the cursors 2.5 divisions (negative and positive) from the trigger point on the first touch. On the second touch, it returns the cursor to its previous position.

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.

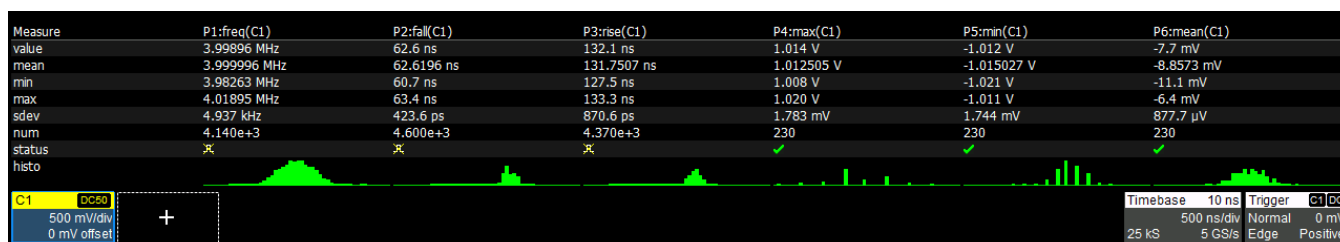
Measure

Parameters are tools that give you access to a wide range of waveform properties, such as Rise Time, RMS voltage and Peak-Peak voltage.

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 math functions, even when the Measure table is hidden from view. The history of a parameter can also be graphed as a [trend](#) for statistical analysis.

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.



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.

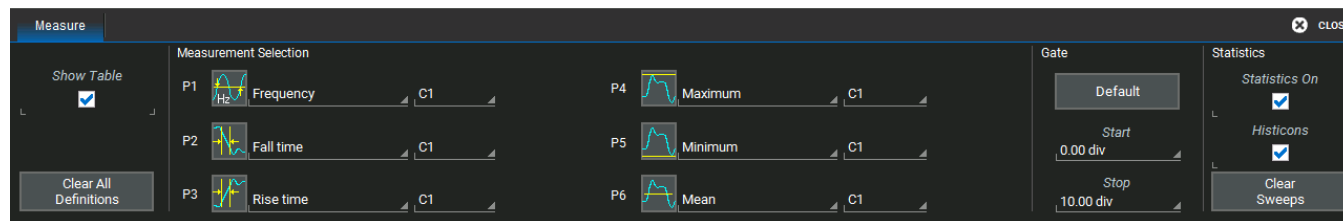
Close setup dialogs when the Measure table is displayed to maximize the area available for viewing waveforms. To return to the Measure dialog when closed, touch anywhere in the table.

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.

Parameter Set Up

The Measure Dialog gives quick access to measurement features. Besides configuring parameters, use the Measure dialog to show [statistics and hysticons](#), or to [gate](#) 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. For each parameter (**Pn**):
 - Touch the **Measure** field and choose a measurement from the list.
 - Touch the **Source** field and choose the source trace to measure. This can be any type of input available to your instrument; all will appear on the Source pop-up selector.
4. Enter any other measurement settings that appear.
5. Optionally:
 - [Gate](#) parameters to limit measurements to only edges inside the gates.
 - Add [Statistics](#) and **Hysticons** 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.

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.

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. You can refine this position to hundredths of a division by using the **Gate Start** and **Stop** fields on the Measure dialog. All parameters share the same gates, and all measurements will change when you drag either gate post to reposition the gate.

Touch **Default** to return the gates to the width of the grid.

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.

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 the Summary field on the Measure dialog.

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, such as triangle or saw-tooth waves, 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 over only the full cycles, rather than the entire acquisition.

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 (such as 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.

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.

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.

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 (such as triangle waveforms), the Top-Base measurement used to calculate the 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 (such as triangle waveforms), the Top-Base measurement used to calculate the amplitude can default to maximum and minimum, giving less predictable results.

Frequency

Reciprocal of each Period of a 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.

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.

Minimum

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

Overshoot-

Amount of overshoot following falling edges, represented as percentage of amplitude. Overshoot- is calculated using the formula $(\text{Base} - \text{Minimum}) / \text{Amplitude} \times 100$. On signals that do not have two major levels (such as triangle 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 $(\text{Maximum} - \text{Top}) / \text{Amplitude} \times 100$. On signals that do not have two major levels (such as triangle or saw-tooth 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.

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 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 (such as triangle waves), the Top-Base measurement used to calculate rise can default to maximum and minimum, giving less predictable results.

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.

Skew

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

Std Dev

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

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

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 ACRMS.

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

Top

Higher vertical value in two-level (bi-modal) signals (the lower is Base), or higher of two most probable waveform states in 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 are not bi-modal (e.g., triangle waves), 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.

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

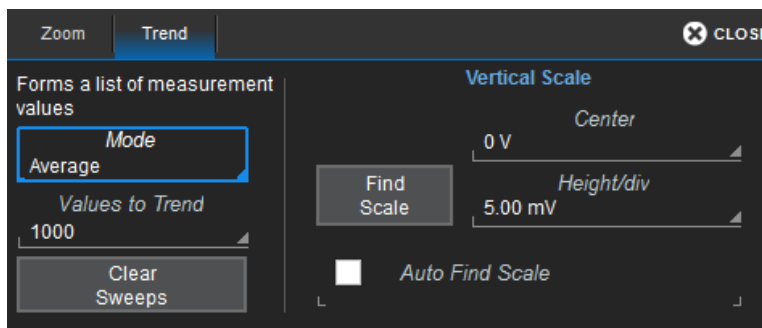
Trends are especially useful for visualizing the history of a parameter over an extended period of time or over multiple acquisitions. Think of Trend as a strip chart recorder for your instrument. Example applications of Trend include:

- [Data logging](#) multiple circuit parameters
- [Power line monitoring](#)
- Measuring [output regulation and ripple](#)

Plotting Trends

Although a Trend plots parameter values, it is created as a Math function on the Function (Fn) dialogs.

1. Select the **Trend Operator** on the Fn setup dialog.



2. Choose a computation **Mode** of All (measurements per acquisition) or Average (one measurement per acquisition).
3. Enter the number of measured **Values to Trend**.

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 F_1 can show the average of C_1 , while trace F_2 provides the integral of F_1).

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

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.

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.

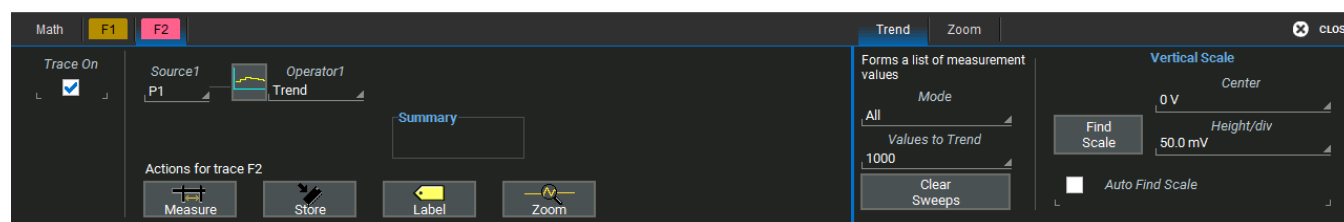
Single functions perform one operation on one or two input sources.

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).



Note: In cases where the math applied results in 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.

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. In **Operator1**, choose the math operation to perform.
3. 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).
4. 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.

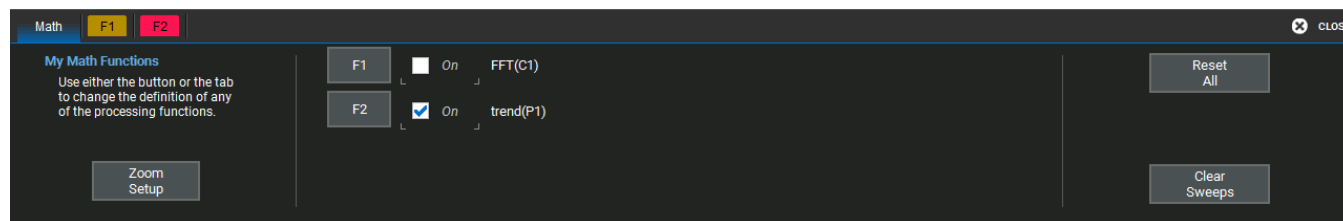
Adjusting Memory or Math Traces

Unlike channel traces, the scale of memory (Mn) or math function (Fn) 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 (Fn) 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 (Zn) in which to draw the trace, but the source trace remains at the original scale.

Math Dialog

Once a math function has been created and saved on the Function (Fn) 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**.

List of Standard Math Operators



Note: The installation of software options on the oscilloscope may add math operators to this list. The keyword to use in remote control programs appears in the Summary field on the Fn dialog.

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.

Derivative

Calculates the derivative of adjacent samples using the formula:

(next sample value – current sample value) / (horizontal sample interval)

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](#).

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.

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:

$(\text{current sample value} + \text{next sample value}) * (\text{horizontal sample interval})$

Each calculated area is summed with the previous sum of areas. The multiplier and adder are applied before the integration function.

Invert

For every point in the waveform, the inverse of that point is calculated.

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.

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: $1 / (\text{sample value})$.

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 values of a waveform at each horizontal value for a specified number of sweeps.

Sinx/x

Performs 10-to-1 interpolation using a $\text{Sin}(x)/x$ filter.

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.

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](#).

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 function](#), 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 (Cr) 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)	$(\text{new data} + 0 * \text{old average}) / (0 + 1) = \text{new data only}$
2	$(\text{new data} + 1 * \text{old average}) / (1 + 1) = 1/2 \text{ new data} + 1/2 \text{ old average}$
3	$(\text{new data} + 2 * \text{old average}) / (2 + 1) = 1/3 \text{ new data} + 2/3 \text{ old average}$
4	$(\text{new data} + 3 * \text{old average}) / (3 + 1) = 1/4 \text{ new data} + 3/4 \text{ old average}$
5	$(\text{new data} + 4 * \text{old average}) / (4 + 1) = 1/5 \text{ new data} + 4/5 \text{ old average}$
6	$(\text{new data} + 4 * \text{old average}) / (4 + 1) = 1/5 \text{ new data} + 4/5 \text{ old average}$
7	$(\text{new data} + 4 * \text{old average}) / (4 + 1) = 1/5 \text{ new data} + 4/5 \text{ old average}$



Note: The number of sweeps used to compute the average is displayed at the bottom of the trace descriptor box.

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.

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.

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 Spectrum, also enter **Line Impedance**. 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.
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 $(\sin X)/X$ 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

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.

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, $V \cdot A$ becomes W).

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

Memory

The instrument is equipped with internal memory slots (Mn) 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.

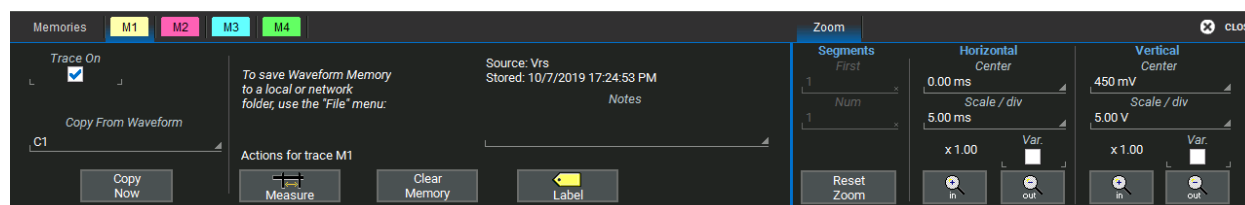
Saving Memories

Store memories on the Memory dialogs (Mn). 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 Mn 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.



Touch the **Add New box** and choose **Memory**. Drag the descriptor box of the trace you wish to store onto the Mn descriptor box.

Or

1. Choose **Math > Memory Setup** to open the Memories dialog.
2. Touch the **Mn 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. Use the Zoom controls to adjust the scale of the memory trace.
6. Optionally, add **Notes** describing the memory.

Save (External) Waveform Files to Memory

Trace (.trc) files saved on other Teledyne LeCroy instruments can also be saved to internal memory. Use the Recall Waveform function to save external files to memory. Then, you can use the Memories dialog to restore them to the touch screen.



Note: The removable MicroSD card serves as internal memory for saving .trc files.

Restoring Memories

The Memories dialog is a convenient panel for restoring saved memories to the display.

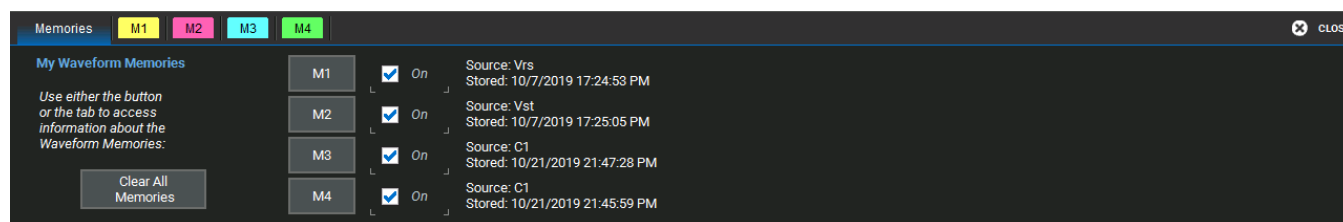
Access the Memories dialog by choosing **Math > Memory Setup**.

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 **Mn** 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 meet mask test limits.

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.

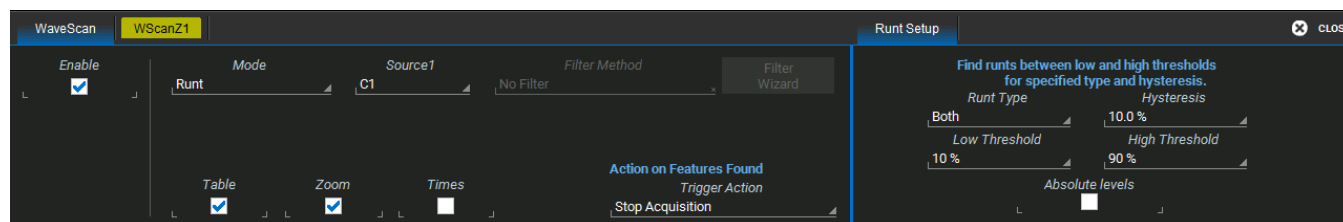
You customize the presentation by choosing different WaveScan displays, called [Scan Views](#). Optionally, set Trigger Actions, such as stopping or beeping, to occur when the scanned events are found.



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. User Action will execute whatever action you have configured for the Save/ User button.

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 threshold. 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.

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(decimal).
- **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.

Additional controls for Zoom view are on the *Zn* dialog. If you turn on a zoom from that dialog, you must turn it 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.

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 *WScanZn* 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.

WaveScan Search

Search is used to find events in traces—usually zoom (*Zn*) traces—that match user-defined criteria. To search within WaveScan:

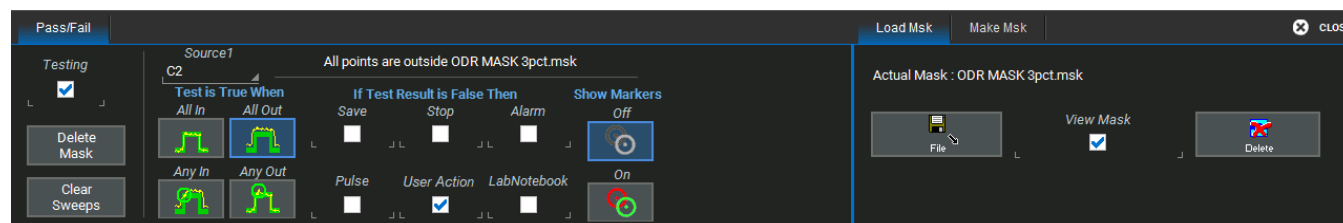
1. Select the **Zoom** view.
2. After stopping the acquisition, open the **WScanZ1 dialog** that appears behind the WaveScan dialog.
3. Use the **Prev** and **Next** buttons to move back or forward within the trace to the events that matched your Scan Modes criteria.

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 is a type of mask testing that is particularly useful for comparing newly acquired signals to a previously acquired "golden standard" waveform. The software counter will update in real time as new acquisitions are compared to the mask, and colored markers will quickly show those areas of the waveform that violate the mask.



Note: You cannot run PASS/FAIL tests on Sequence Mode acquisitions.

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.

Access Mask Test Dialogs

Choose **Analysis > Pass/ Fail Setup** to display the **Pass/ Fail** dialog. To the right are the **Load Mask** and **Make Mask** subdialogs where you make and manage masks.

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

Use this procedure in lieu of Make Mask if you have a pre-defined mask file.

1. Open the **Load Mask** subdialog.
2. 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 Mask** button at the left of the Pass/Fail dialog.

Run Test

1. On the main section of the Pass/Fail dialog, select the condition of the signal that, when "True", constitutes a "Pass" (samples are **All In**(side mask), **All Out**(side mask), etc.).
2. Select any actions to take when the test produces this result:
 - **Save** a waveform file
 - **Stop** the test
 - Sound an **Alarm**
 - Emit a **Pulse** from Aux Out(put)
 - Capture the screen and process it according to your **User Action** setting
 - Create a **LabNotebook** Entry
3. Choose to turn **Markers Off** or **On**. When "On", points where the signal intersects the mask are marked by a red circle.
4. Check the **Testing** box at the far left of the Pass/Fail dialog. The results of your test will appear in a table below the grid as soon as there is a fresh acquisition.

Saving Data (File Functions)

The File menu functions allow you to save and recall setups, waveform data, table data, screen captures, and LabNotebooks. 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. 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.

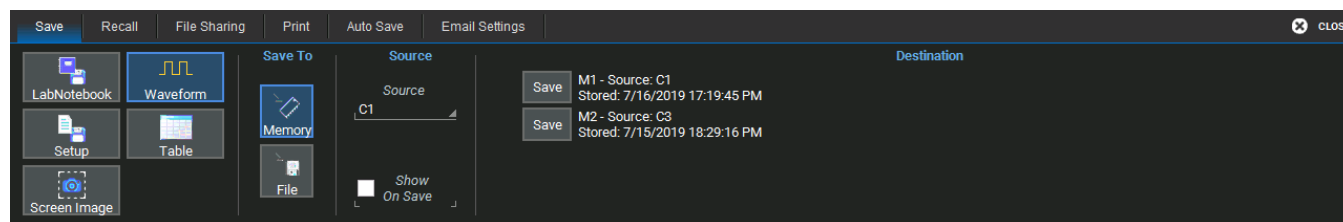
Save Waveform

The Save Waveform function saves trace data either to internal memory (M1-Mn) 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.



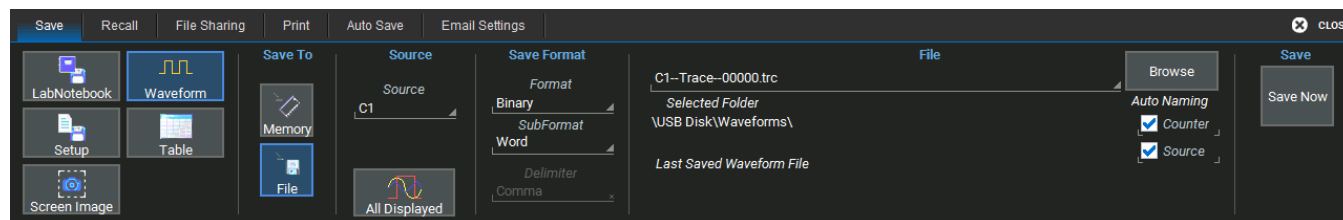
Note: Only traces saved in Teledyne LeCroy proprietary formats can be recalled to the display. Use the Binary format (.trc) for analog waveforms, or WaveML (.xml) for digital waveforms.

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

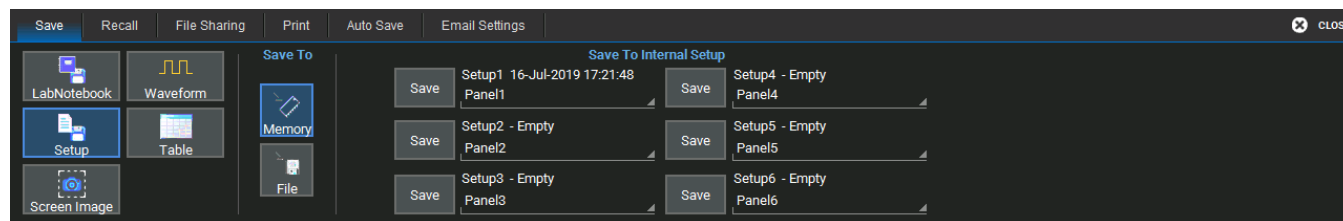


1. From the menu bar, choose **File > Save Waveform**.
2. Choose to Save To **File**.
3. Choose the **Source** waveform. To quickly save all displayed waveforms separate trace files, touch the **All Displayed** button.
4. Select a file **Format**. **Binary** is Teledyne LeCroy's .trc file format. Binary results in the smallest possible file size. Binary files can be converted to ASCII using Teledyne LeCroy utilities such as WaveStudio.
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

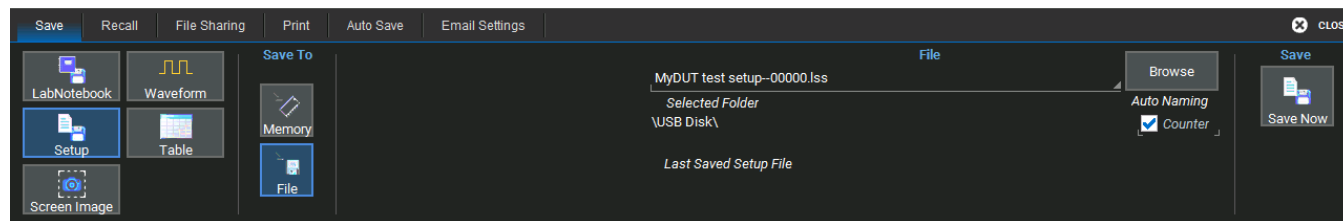
You can quickly save oscilloscope configurations to one of the internal setup panels or to a LeCroy System Setup (.lss) file, a text-based Automation program. Setups can be easily [recalled](#) to restore the oscilloscope to the saved state.

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 **Paneln**. 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.

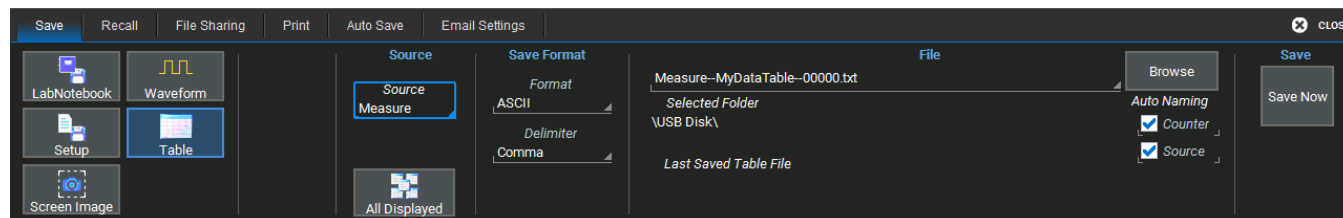
Save Setup to File



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.

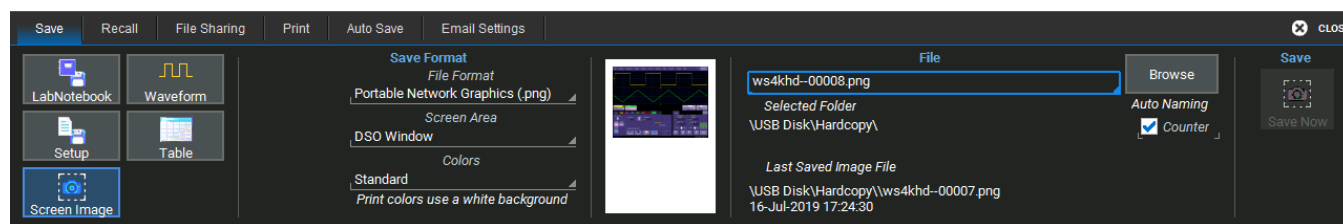


1. From the menu bar, choose **File > Save**, then choose **Table**.
2. To save only one of the tables displayed, touch **Source** and navigate to the selection.

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.

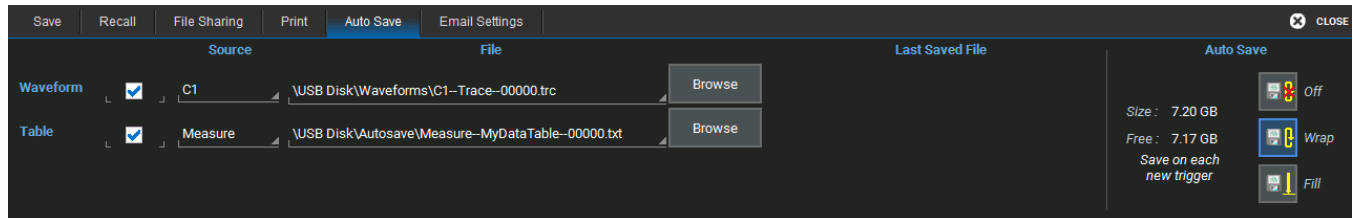


1. From the menu bar, choose **File > Save**, then choose **Screen Image**.
2. Choose a **File Format** of .JPEG, .PNG, .TIF, or .BMP.
3. Select the **Screen Area** captured:
 - Grid Area Only includes any visible waveform grids, tables, and descriptor boxes.
 - DSO Window includes the above plus any open dialogs, menu bar, and message bar.
 - Full Screen is the full Windows display, including other visible applications and desktop.
4. 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.
5. 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.
6. Touch **Save Now**.

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 Micro SD card. 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.

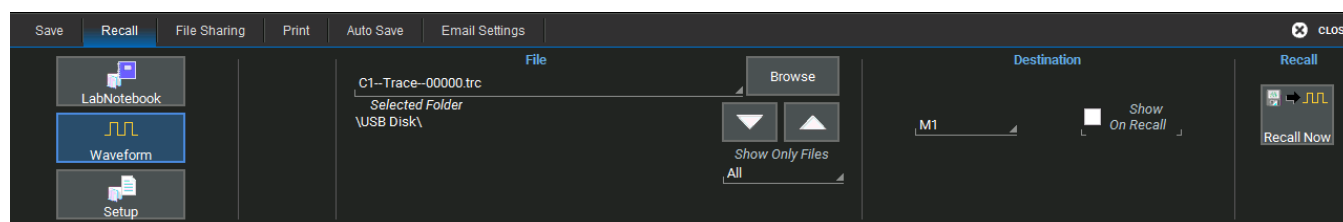
Waveform data stored to .TRC or .XML files can be recalled into a memory and from there restored to the screen.

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.

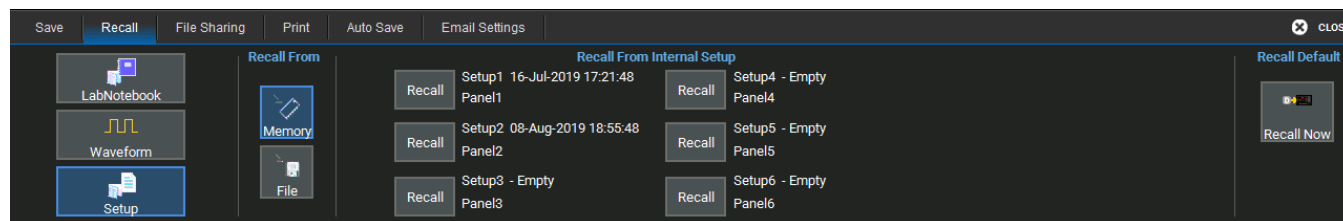
Optionally, touch **Show only files** to apply a search filter (channels, math functions, or memory) to the list of available files.

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

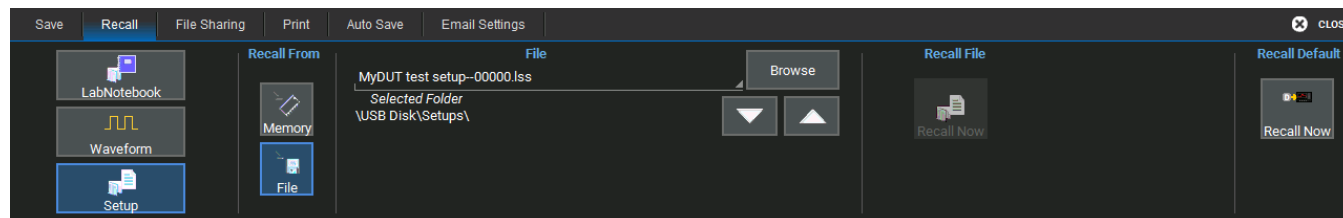


1. Choose to Recall From **Memory**.
2. Touch one of the six **Recall** buttons under **Recall From Internal Setup...**



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

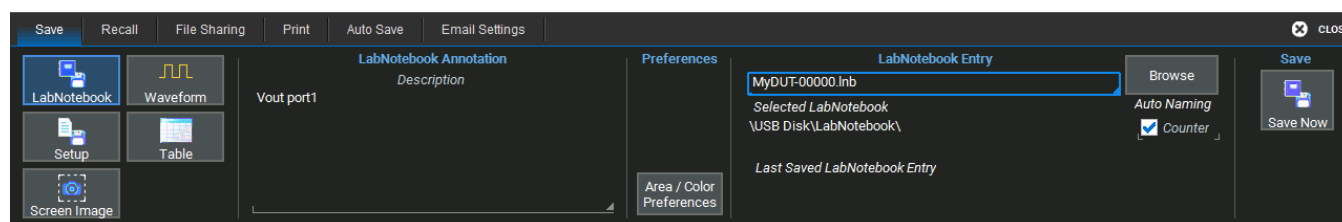
The LabNotebook feature allows you to create and save composite files containing a screen capture of all displayed waveforms, as well as all waveform and setup data at the time of capture. The [Flashback](#) feature instantly recalls the setups and waveforms stored with LabNotebook Entries, enabling you to restore the exact state of the instrument at a later date to perform additional analysis.

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.



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.
3. 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 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.

4. 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 Micro SD card.



Note: Changing the file path, name or format will reset the counter to the next number in that sequence.

5. If you do not want to use autonumbering to identify files, deselect **Counter**.
6. Touch **Save Now**.

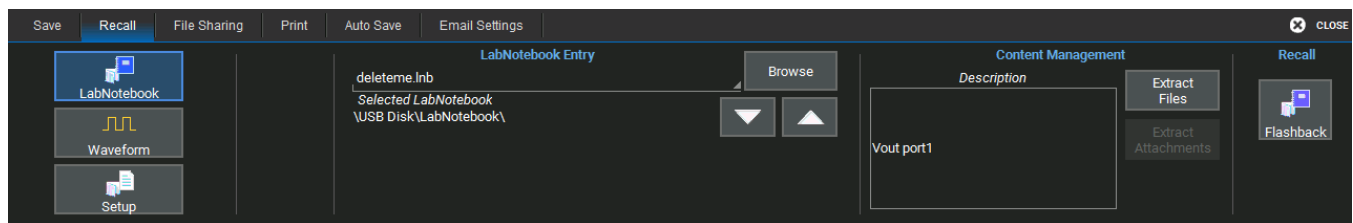
Recall LabNotebook

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.



Note: Conversion from legacy LabNotebook .ZIP format is not supported. The file must be created in the new .LNB format to be recalled.

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 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.

Extract Files

The component files that make up the LabNotebook composite (.PNG, .LSS, and .TRC) 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.

Some result data *not* included in the Flashback are:

- **Persistence data**, although it is saved with the .LNB file.
- **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.

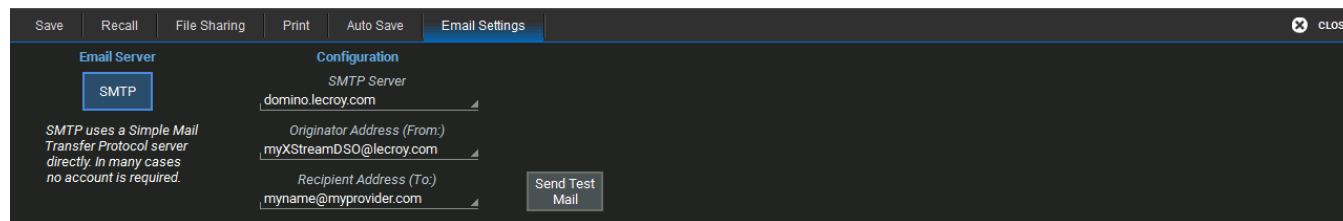
Share

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**.

Email Settings

Configure the oscilloscope for email on the Email Settings dialog.

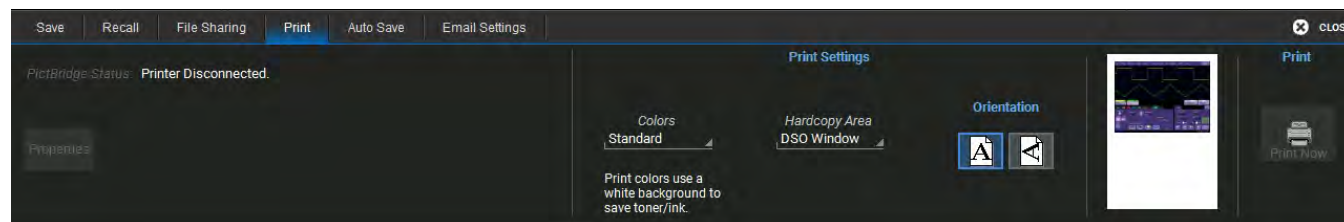


1. Select to use **MAPI** or **SMTP**.
2. If you chose SMTP, touch **SMTP Server** and enter the network address of your mail server.
3. Touch **Originator Address (From:)** and enter the instrument's e-mail address.
4. Touch **Default Recipient Address (To:)** and enter the recipient's e-mail address.
5. Use **Send Test Mail** to send a confirmation message to ensure proper e-mail configuration.

Print

Print captures an image of the touch screen display and sends it to a printer formatted according to your selections here. These settings also apply wherever you have a Print Screen option on a dialog.

The **Print Now** button at the far right of the dialog can be used to execute your current Print selection.

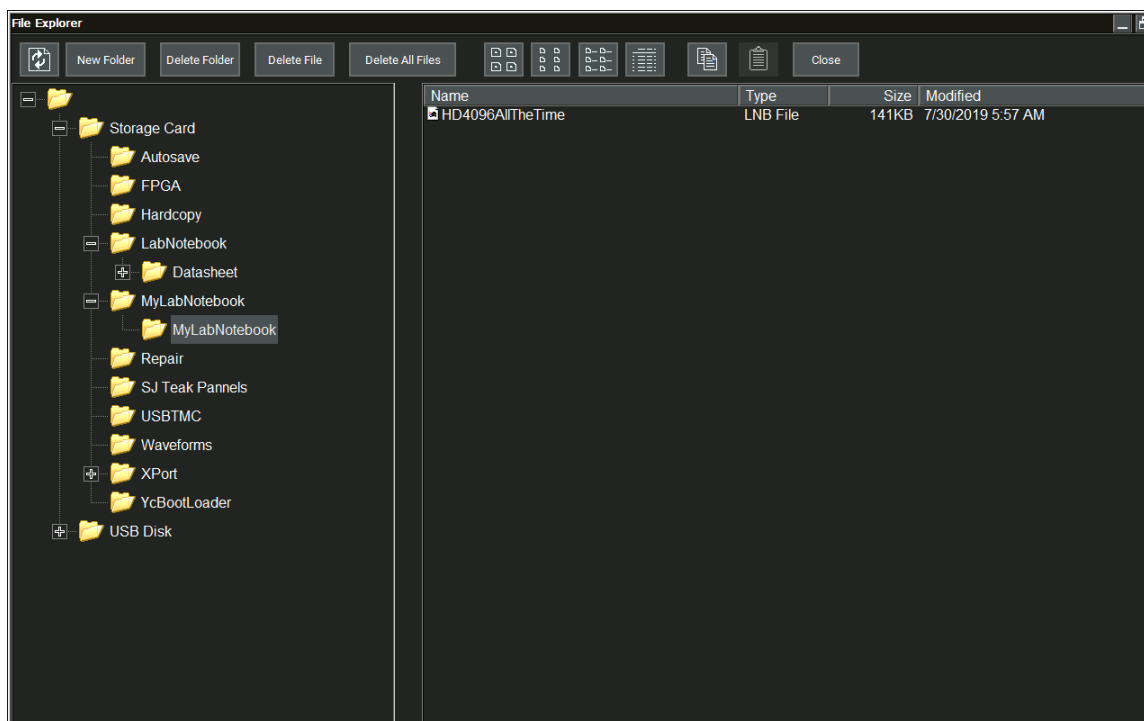


Note: Only Pictbridge printers with a USB connection to the oscilloscope are supported.

1. Choose the **Colors** used:
 - Standard uses a black background as it appears on the instrument.
 - Print uses a white background.
 - Black & White captures the image in grayscale.
2. Select a **Hardcopy (Print) 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.
3. Choose a paper orientation of **Portrait** or **Landscape**.
4. To modify any other printer settings, touch **Properties**.

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).

To copy a folder or file, select it and touch the **Copy** icon . Navigate to the desired directory and touch the **Paste** icon .

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 SD card 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-CANFDTTable-..." will reset the counter to "Decode1-CANFDTTable-0000" if there is no prior file named "Decode1-CANFDTTable-...".

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.

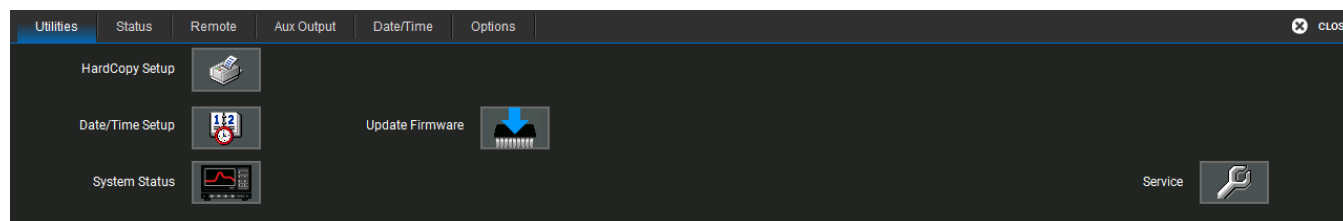
Utilities

[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.

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.

Update Firmware opens a dialog for [installing the oscilloscope firmware](#).

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.

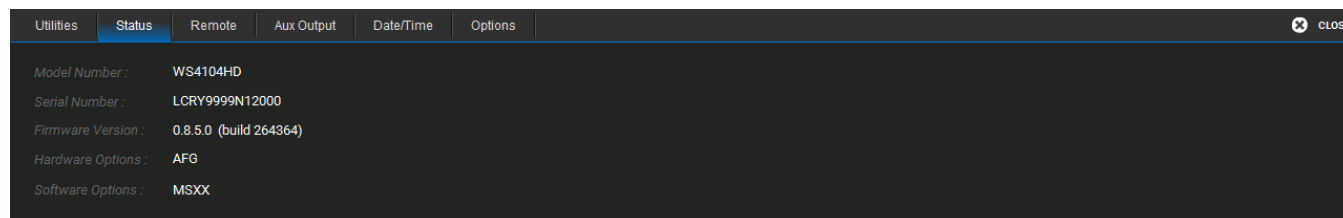
Status

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 for remote control, also install Teledyne LeCroy's VICP drivers on the controller. These are included in the VICP Passport plug-in,

The instrument uses Dynamic Host Configuration Protocol (DHCP) as its default addressing protocol, although you can manually assign a static IP address.

- **USBTMC.** Connect a cable to the USBTMC port on the back of the oscilloscope.
- **GPIOB.** To activate this option, connect the USB2-GPIB adapter to any host USB port.



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.

Assign Static IP Address/Name Server

Before starting, consult with your Network Administrator regarding the oscilloscope's network address, subnet, default gateway.



Note: You can also use this procedure to assign a name server if your network uses DHCP addressing.

1. Connect a keyboard to the front panel USB port.
2. From the menu bar, choose **Utilities > Utilities Setup**, then touch the **Remote tab**.
3. On the **Remote** dialog, touch **Net Connections**.
4. Touch the icon to the right of the Make New Connections icon.
5. On the Ethernet Driver Settings dialog, choose **Specify an IP Address** if setting a static IP; leave Obtain an IP address via DHCP if assigning a Name Server.
6. Touch the **IP Address** field, and use the keyboard to enter the address. Repeat for Subnet Mask and Default Gateway.
Alternatively, touch the Name Server tab and enter the DNS server address.
7. Touch the window close boxes to return to the oscilloscope application.

Enter Network ID

The network file sharing capabilities require that the oscilloscope have access to the network domain and shares. Use the Network ID settings to enter network credentials. Once credentials have been saved, the oscilloscope should connect to the network seamlessly; you do not need to re-enter credentials unless you wish to change them.

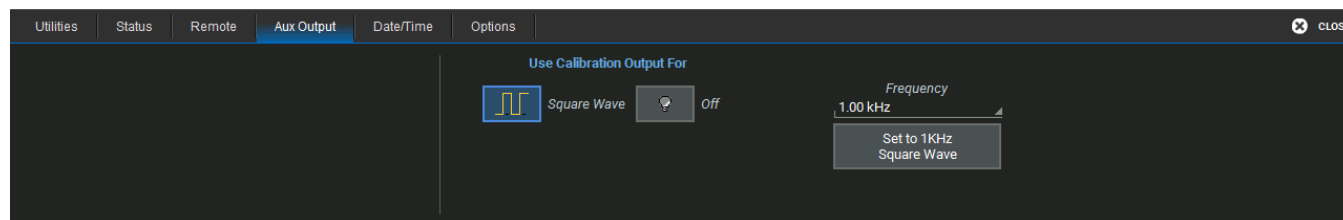


Note: These may be the credentials of any valid domain user (e.g., your own login); they do not have to be unique to the oscilloscope. Consult with your Network Administrator.

1. Connect a keyboard to the front panel USB port.
2. From the menu bar, choose **Utilities > Utilities Setup**, then touch the **Remote** tab.
3. On the Remote dialog, touch **Network ID**.
4. On the Owner Properties dialog, enter the network **User Name**, **Password**, and **Domain Name**.
5. Reboot the oscilloscope after entering or changing the Network ID.

Aux Output

Use the Aux Output dialog to configure the frequency of the signal output from the **Cal Out** hook on the front of the oscilloscope. The signal is a 3 V square wave that may be used to compensate probes.



To turn on the calibration signal:

1. Under **Use Calibration Output For**, choose **Square** wave.
2. Either enter the **Frequency**, or use the **Set to 1 KHz Square Wave** button.

Touch **Off** to disable the calibration signal.



Note: The **Aux Out** port on the back of the oscilloscope outputs a 3.3 V TTL pulse to another device following a trigger enabled or Pass/ Fail event when the Pulse Aux Out action is selected.

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.

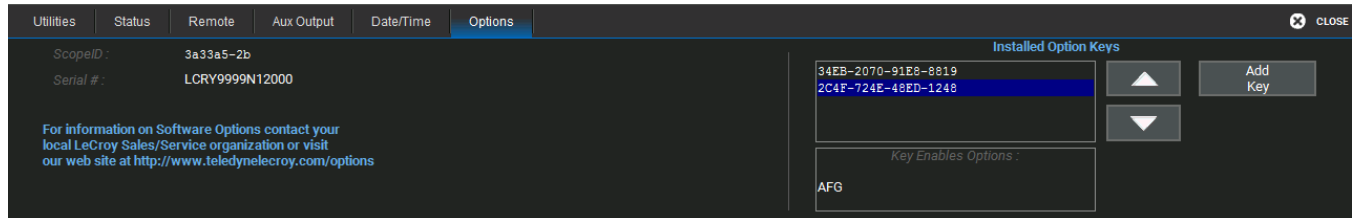
Time Zone

Select a **Time Zone**.

Check **Auto Daylight Savings Time** to automatically reset the timestamp when daylight savings occurs in the selected time zone.

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 **Utilities > 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.

Access the **Disk Utilities** dialog by selecting **Utilities > Disk Utilities** from the menu bar.

Using the File Browser

Touch **Launch File Browser** to display the same dialog as used for managing files when saving/recalling waveforms, setups or LabNotebooks. See [Using the File Browser](#).

Backing Up Files

Touch **Backup** to back up the entire contents of the internal storage to a removable storage device. If a USB drive is installed, you can choose to back up to USB or to the removable Storage Card (Micro SD Card). Otherwise, Storage Card is the default.

Sanitizing the Instrument

Touch **Sanitize** to delete all data from the oscilloscope internal storage (stored setup panels, memories, waveform data, measurement and pass/fail data, logs, etc.), returning the oscilloscope to a "clean" state without necessarily reverting the oscilloscope to a factory default state.

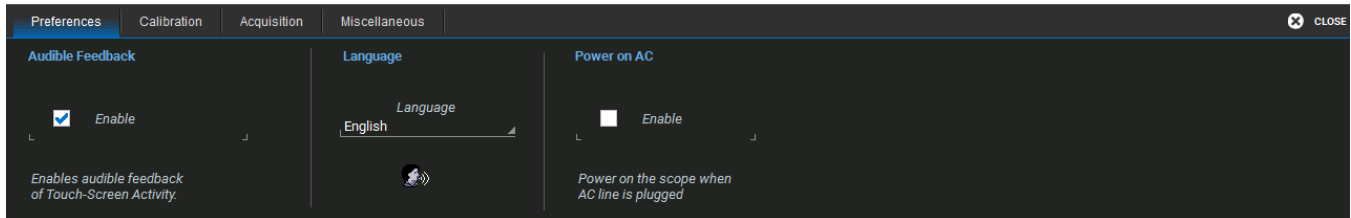


Note: The Sanitize function does not affect files stored on the Micro SD Card (D: drive). The card can be removed before or after sanitizing for storage in a secure facility.

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.

Calibration

To ensure the instrument maintains specified performance, it is factory set to perform a calibration during warm-up. We recommend that you warm up the oscilloscope for at least 20 minutes prior to use to give the instrument time to complete calibration procedures.

Manually calibrate the oscilloscope when:

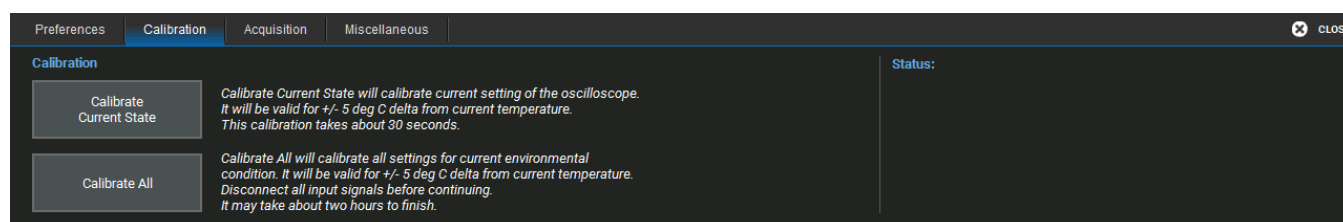
- It is used in temperatures that differ from the previous calibration temperature by more than 5° C.

OR

- It has been more than one month since the previous calibration.

Go to **Utilities > Calibration Setup**.

The Calibration dialog shows the oscilloscope's calibration status and recommended actions.



There are two options for this calibration:

Calibrate Current Setting—the oscilloscope is calibrated at the current vertical and horizontal setting. This calibration is valid for this setting for the current temperature $\pm 5^{\circ}$ C and takes under 30 seconds.

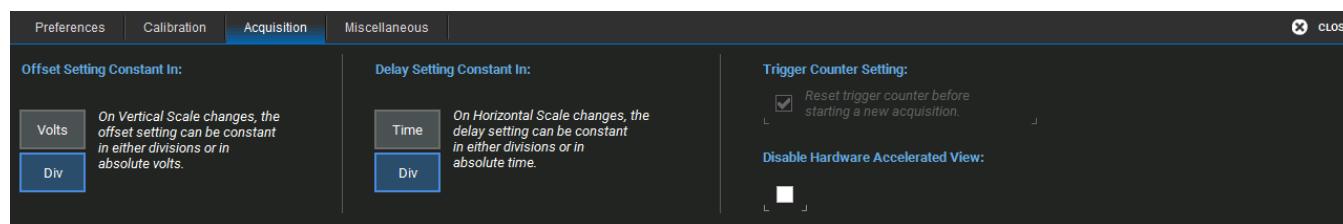
Calibrate All—all possible combinations of vertical and horizontal settings are calibrated at the current temperature. This calibration is valid for the current temperature $\pm 5^{\circ}$ C and takes about 120 minutes.



Caution: It is required that all inputs be removed from the oscilloscope prior to performing calibration.

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:

- **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.



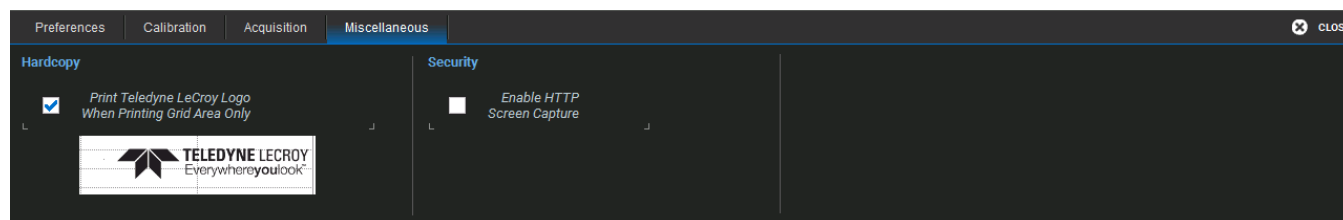
Note: Whenever Div is selected, values are scaled proportional to the change in gain or timebase, thereby keeping the division of the grid constant.

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.

Select **Disable Hardware Accelerated View** to disable the fast update of the display.

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.

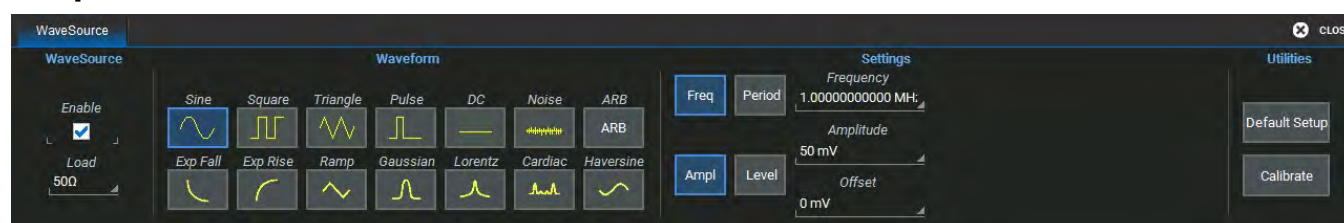
Check **Enable HTTP Screen Capture** to enable remote capture of the touch screen display over a network. This setting is required to use the instrument with the WaveStudio software.

WaveSource Automatic Waveform Generator

The WaveSource Automatic Waveform Generator allows you to output custom sine, square, triangle, pulse, DC, noise, and arbitrary waveforms from the oscilloscope.

Connect a BNC cable from the WaveSource output on the back of the oscilloscope to the external device. Output is continuous once WaveSource is enabled.

Output Standard Waveform



1. Choose **Utilities > WaveSource** or touch the front panel **WaveSource** button.
2. Select the **Load** level.
3. Choose a **Waveform** type.
4. Depending on the type, waveforms may be configured using different groups of **Settings** (e.g., Frequency or Period, but not both). When you see buttons offering this choice, first select the group of settings to use.
5. Adjust parameters (e.g., Frequency) by touching the field and turning the front panel Adjust knob, or touch twice and use the soft keypad to enter new values.

Touch **Default Setup** to restore the default parameters.

6. Check **Enable** to begin output.

Import and Output Arbitrary Waveform

The Arbitrary waveform feature enables you to import waveforms via file, then output this waveform from the function generator.

Waveform files must be saved in .csv format (using Excel®, WaveStation™, etc.) to be imported into WaveSource. Use the [Save Waveform](#) file function to save waveforms from the oscilloscope for output via WaveSource.

To import an external file, first save it to a USB drive or transfer it directly to the WaveSurfer storage card if your oscilloscope is networked.

1. Choose **Utilities > WaveSource** or touch the front panel **WaveSource** button.
2. Select the **Load** level.
3. Choose **Waveform** type **ARB**.

4. Touch **Browse** next to the File Name field and browse to the location of the arbitrary waveform file on either the Storage Card or the USB Disk. Select the file and touch **OK**.
5. Touch **Upload**.
6. Check **Enable** to begin output.



Note: When outputting Arbitrary waveforms, the Frequency shown on the WaveSource dialog is the pattern repetition rate, not the actual Frequency of the source waveform from which the waveform file was created.

Calibrate Function Generator

Touch the **Calibrate** button to start the internal calibration of the Function Generator.



Caution: Remove all cables before starting the calibration.

Digital Voltmeter

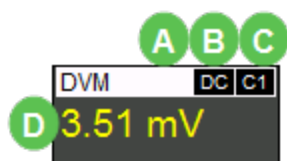
The Digital Voltmeter option activates an integrated 4-digit digital voltmeter and 5-digit frequency counter that operates through the same probes already attached to the oscilloscope channels. Use it to view real-time measurements through a dedicated user interface display that continues even when your triggering system is stopped.



Note: You do not need to display the source channel trace to see the voltage readout on the DVM descriptor box and dialog.

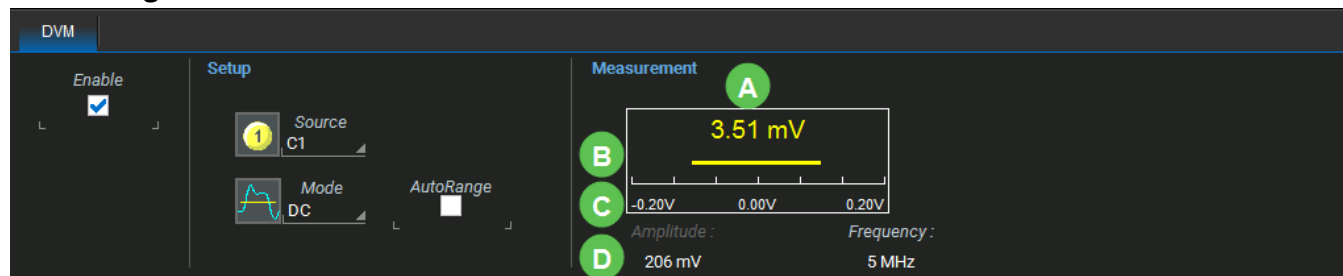
DVM Readout

DVM Descriptor Box



- A. AutoScale indicator; "A" when on, blank when off
- B. Mode
- C. Input channel measured
- D. Voltage readout, color coded to match input channel

DVM Dialog



- A. Last measured voltage
- B. Histogram of measured voltage
- C. Vertical range (in -/+ divisions) of signal
- D. Amplitude and Frequency readout

Set Up DVM

1. From the menu bar, choose **Utilities > DVM**.
2. Mark **Enable** to turn on the DVM function.
3. Touch **Source** and select the input channel to be measured.
4. Touch **Mode** and choose the waveform parameter to display on the readout. Options are:
 - DC
 - DCRMS
 - ACRMS
 - Frequency
5. Optionally, turn on **AutoRange**, which automatically adjusts the Vertical range and offset of the waveform as the amplitude changes so that measurements are always taken at the optimal resolution. When this option is selected, the letter A appears on the DVM descriptor readout.

Maintenance

Topics in this section describe procedures for keeping the instrument in optimal working condition.

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**.

Firmware Update for CE Oscilloscopes

This procedure is for oscilloscopes running the Microsoft Compact Embedded (CE) operating system.

Teledyne LeCroy frequently releases free firmware updates containing new product features and bug fixes. The installer updates multiple components including the oscilloscope application, required DLLs, drivers, and low-level microcode for integrated circuits on the oscilloscope.

The firmware update procedure *does not* modify or delete any user data (setups, waveforms, screen captures, calibration constants, etc.) stored on the oscilloscope MicroSD card.

1. Visit **Oscilloscope Downloads > Firmware Upgrades**.
2. Select your oscilloscope **series** and **model number**.
3. Enter your registration **login** information, or create a new account.
4. Click the **download** link, and choose to **Save** the installer to a USB storage device.
5. Insert the USB device into one of the ports on the front of the oscilloscope.
6. Go to **Utilities > Utilities Setup** and choose **Update Firmware**.
7. Browse to the installer file in the USB Disk folder, then click **Upgrade**.
8. When installation is complete, choose **Reboot Now**.



Caution: The installation may take several minutes, depending on the length of time since your last upgrade. **Do not power down the oscilloscope at any point during the installation process.**

Release Notes

Release Notes are provided for each new version. They can be found on the series download page where you downloaded the installer.

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 planes⁴

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 lines⁴

EN 61000-4-5:2014+A1:2017, Power Line Surge, 1 kV AC Mains L-N, L-PE, N-PE⁴

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.

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

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



ISO Certification

Manufactured under an ISO 9000 Registered Quality Management System.

Intellectual Property

All patents pertaining to the WaveSurfer 4000HD can be found on our website at:

Warranty

THE WARRANTY BELOW REPLACES ALL OTHER WARRANTIES, EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTY OF MERCHANTABILITY, FITNESS, OR ADEQUACY FOR ANY PARTICULAR PURPOSE OR USE. TELEDYNE LECROY SHALL NOT BE LIABLE FOR ANY SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES, WHETHER IN CONTRACT OR OTHERWISE. THE CUSTOMER IS RESPONSIBLE FOR THE TRANSPORTATION AND INSURANCE CHARGES FOR THE RETURN OF PRODUCTS TO THE SERVICE FACILITY. TELEDYNE LECROY WILL RETURN ALL PRODUCTS UNDER WARRANTY WITH TRANSPORT PREPAID.

The product is warranted for normal use and operation, within specifications, for a period of three years from shipment. Teledyne LeCroy will either repair or, at our option, replace any product returned to one of our authorized service centers within this period. However, in order to do this we must first examine the product and find that it is defective due to workmanship or materials and not due to misuse, neglect, accident, or abnormal conditions or operation.

The instrument's firmware has been thoroughly tested and is presumed to be functional. Nevertheless, it is supplied without warranty of any kind covering detailed performance.

Teledyne LeCroy shall not be responsible for any defect, damage, or failure caused by any of the following:

a) attempted repairs or installations by personnel other than Teledyne LeCroy representatives or b) improper connection to incompatible equipment, or c) for any damage or malfunction caused by the use of non-Teledyne LeCroy supplies. Furthermore, Teledyne LeCroy shall not be obligated to service a product that has been modified or integrated where the modification or integration increases the task duration or difficulty of servicing the instrument. Spare and replacement parts, and repairs, all have a 90-day warranty.

Products not made by Teledyne LeCroy are covered solely by the warranty of the original equipment manufacturer.