



Operator's Manual

WaveSurfer 3000/3000z Oscilloscopes

Contents

Safety	1
Symbols	1
Precautions	1
Operating Environment	2
Cooling	2
Cleaning	2
Power	3
Oscilloscope Overview	5
Front Input/Output Panel	5
Back Input/Output Panel	5
Front Panel	6
Signal Interfaces	8
Micro SD Card	10
Oscilloscope Set Up	11
Powering On/Off	11
Software Activation	11
Positioning the Feet	11
Connecting to Other Devices/Systems	12
Language Selection	13
Auto Setup	13
Recall Default Setup	13
Using MAUI	15
Touch Screen	15
Touch Actions	21
Working With Traces	22
Zooming	25
Print/Screen Capture	28
Acquisition	29
Vertical	29
Digital (Mixed Signal)	32
Timebase	37
Trigger	44
Viewing Status	53
Display	55
Display Set Up	55
Persistence Display	57

Math and Measure	59
Cursors	59
Measure	62
Math	70
Memory	83
Analysis Tools	85
WaveScan	85
Pass/Fail Testing	89
Save / Recall	91
Save Setups	91
Recall Setups	92
Save Waveforms	93
Recall Waveforms	95
Save Table Data	96
Auto Save	97
LabNotebook	98
Utilities	101
Utilities Dialogs	101
Disk Utilities	108
Preferences Dialogs	110
WaveSource Automatic Waveform Generator	115
Digital Voltmeter	117
Maintenance	119
Fuse Replacement	119
Touch Screen Calibration	119
Restart/Reboot Instrument	119
WaveSurfer 3000 Firmware Update	120
Technical Support	121
Returning a Product for Service	122
Certifications	123
EMC Compliance	123
Safety Compliance	124
Environmental Compliance	125
ISO Certification	125
Warranty	126
Intellectual Property	126
Index	127

Safety

To maintain the instrument in a correct and safe condition, observe generally accepted safety procedures in addition to the precautions specified in this section. **The overall safety of any system incorporating this product is the responsibility of the assembler of the system.**

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. Do not proceed until the information is fully understood and conditions are met.



Caution, high voltage; risk of electric shock or burn.



Frame or chassis terminal (ground connection).



Alternating current.



Standby power (front of instrument).

Precautions



Caution: Comply with the following to avoid personal injury or damage to your equipment.

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

Maintain ground. This product is grounded through the power cord grounding conductor. To avoid electric shock, connect only to a grounded mating outlet.

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 power cord shipped with this 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. Obviously incorrect measurement behaviors (such as failure to calibrate) might indicate hazardous live electrical quantities. Cease operation immediately and secure the instrument from inadvertent use.

Operating Environment

Temperature: 0 to 50° C.

Humidity: Maximum relative humidity 90 % for temperatures up to 31° C, decreasing linearly to 50% relative humidity at 40° C.

Altitude: Up to 3,000 m at or below 30° C.

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 the instrument thoroughly before connecting a live voltage source.



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

Power

The instrument operates from a single-phase, 100 to 240 Vrms ($\pm 10\%$) AC power source at 50/60 Hz ($\pm 5\%$), or a 100 to 120 Vrms ($\pm 10\%$) AC power source at 400 Hz ($\pm 5\%$). The instrument automatically adapts to the line voltage. Manual voltage selection is not required.

Power Consumption

Power Consumption	WaveSurfer 3000z	WaveSurfer 3000	
	4-channel	4-channel	2-channel
Nominal	80 W (80 VA)	80 W (80 VA)	65 W (65 VA)
Maximum*	150 W (150 VA)	150 W (150 VA)	100 W (100 VA)
Standby	4 W	4 W	4 W

* Maximum consumption represents power accessories installed on all inputs/outputs (e.g., active probes, USB peripherals, digital leadset, etc.).

Ground

The AC inlet ground is connected directly to the frame of the instrument. For adequate protection against electric shock, connect to a mating outlet with a safety ground contact.



Caution: Only use the power cord provided with your instrument. Interrupting the protective conductor inside or outside the oscilloscope, or disconnecting the safety ground terminal, creates a hazardous situation. Intentional interruption is prohibited.

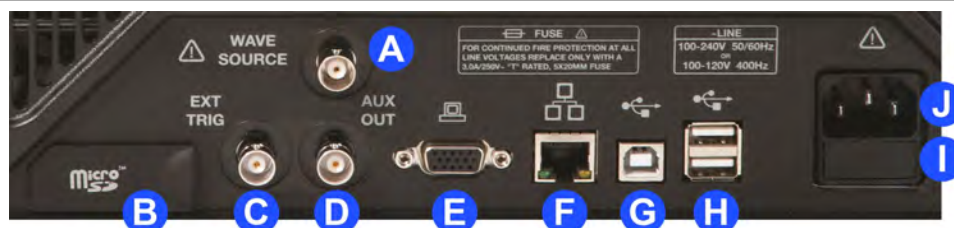
Oscilloscope Overview

Front Input/Output Panel



- A. Power button.
- B. Channel inputs 1-4 for analog signals.
- C. Mixed signal interface for digital inputs (WS3K-MSO required).
- D. Front-mounted host USB port for transferring data or connecting peripherals such as a mouse or keyboard.
- E. Ground and calibration output terminal used to compensate passive probes.

Back Input/Output Panel



- A. WaveSource connector outputs signal from the internal waveform generator.
- B. MicroSD Card slot.
- C. EXT Trig connector accepts external trigger.
- D. AUX OUT connector sends trigger out.
- E. VGA connector sends video out to external monitors.
- F. Ethernet port connects the oscilloscope to a LAN.
- G. USBTMC port enables remote control of the oscilloscope.
- H. Additional host USB ports (2) connect external devices such as printers or storage drives.
- I. Fuse holder.
- J. AC Power inlet.

See the general set up instructions for more information about configuring [connections to other devices](#).

Front Panel

The Front Panel houses "hard" controls for basic oscilloscope functions. See the later sections of this manual for instructions on using the touch screen to make the settings described here.



All the knobs on the front panel function one way if turned and another if pushed like a button. The top label describes the knob's "turn" action, the bottom label its "push" action.

Front panel buttons light up to indicate which traces and functions are active. Actions performed from the front panel always apply to the active trace.

Miscellaneous Controls

Auto Setup performs an [Auto Setup](#).

Default Setup resets the oscilloscope to the factory defaults.

Print captures the entire screen and outputs it according to your [Hardcopy settings](#).

Touch Screen enables/disables touch screen functionality.

Clear Sweeps resets the acquisition counter and any cumulative measurements.

Shortcut Buttons

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

WaveScan opens the WaveScan dialog.

History opens the History Mode dialog.

WaveSource opens the WaveSource waveform generator dialog if you have the function generator option installed.

Trigger Controls

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

READY indicator lights when the trigger is armed. **TRIG'D** is lit momentarily when a trigger occurs. A fast trigger rate causes the light to stay lit continuously.

Setup corresponds to the menu selection Trigger > Trigger Setup. Press it once to open the Trigger Setup dialog and again to close the dialog.

Auto turns on Auto trigger mode. The oscilloscope triggers after a time-out, even if the trigger conditions are not met.

Normal turns on Normal trigger mode. The oscilloscope triggers each time a signal is present that meets the conditions set for the type of trigger selected.

Single turns on Single trigger mode. The oscilloscope triggers once (single-shot acquisition) when the input signal meets the trigger conditions. If the oscilloscope is already armed, it will force a trigger.

Stop prevents the oscilloscope from triggering on a signal. If you boot up the instrument with the trigger in Stop mode, a "No trace available" message is shown. Press the Auto button to display a trace.

Horizontal Controls

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

The **Horizontal Adjust knob** sets the Time/division (S) of the oscilloscope acquisition system when the trace source is an input channel. The Time/div value is shown on the Timebase descriptor box. When using this control, the oscilloscope allocates memory as needed to maintain the highest sample rate possible for the timebase setting. When the trace is a zoom, memory or math function, turn the knob to change the horizontal scale of the trace, effectively "zooming" in or out. By default, the knob adjusts values in 1, 2, 5, 10 step increments. Push the knob to change the action to fine increments; push it again to return to stepped increments.

Vertical Controls

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

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.

Gain knob sets Vertical Gain (V/div). The value appears on the trace descriptor box. By default, the knob adjusts values in 1, 2, 5, 10 step increments. Push the knob to change the action to fine increments; push it again to return to stepped increments.

Dig button enables digital input through the Digital Leadset on -MS models.

Math, Zoom, and Mem(ory) Buttons

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

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

Cursor Controls

Cursors identify specific voltage and time values on the waveform. The white cursor lines help make these points more visible. A readout of the values appears on the trace descriptor box.

There are five preset cursor types, each with a unique appearance on the display. These are described in more detail in the [Cursors](#) section.

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

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

Adjust and Intensity Controls

The **Adjust knob** changes the value in any highlighted data entry field when turned. Pushing the Adjust knob toggles between coarse (large increment) or fine (small increment) adjustments.

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

Signal Interfaces

MAUI instruments offer a variety of interfaces for using probes or other devices to input analog or digital signals. See the oscilloscope product page at

Analog Inputs

A series of connectors arranged on the front of the instrument are used to input analog signals on channels 1-4. EXT on the back can be used to input an external trigger pulse.

WaveSurfer channel connectors use the ProBus interface. The ProBus interface incorporates a 6-pin power and communication connection and a BNC signal connection to the probe. It includes sense rings for detecting passive probes and accepts a BNC cable connected directly to it. ProBus offers 50 Ω and 1 M Ω input impedance and control for a wide range of probes.

The channel interfaces power probes and completely integrate 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.

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.

Active Probes

Teledyne LeCroy offers a variety of active probes for use with your oscilloscope:

- **Single-Ended** active probes can measure voltages at high frequencies. Use of an active probe requires a test point and a ground point. The ground (also called earth) acts as a zero reference for the test point measurement.
- **Differential Probes** are like two probes in one. Instead of measuring a test point in relation to a ground point, differential probes measure the difference in voltage of two test points.

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.

Digital Leadset



The digital leadset provided with Mixed Signal model oscilloscopes enables input of up-to-16 lines of digital data. Physical lines can be preconfigured into different logical groups corresponding to a bus and renamed appropriately depending on the group.

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 numbered the same as the physical lead they represent, but any line number can be re-assigned to any lead on the Digital setup dialogs.

Connecting/Disconnecting the Leadset

The digital leadset connects to the Mixed Signal interface on the front of the instrument.

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.

Grounding Leads

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.

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 trace 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.

Oscilloscope Set Up

Powering On/Off



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

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 for 7 seconds will execute a “hard” shut down (as on a computer), which we do not recommend doing because it does not allow the Windows 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.

Software Activation

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

Firmware

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 on the website to download and install the software.

Purchased Options

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.

Positioning the Feet

The WaveSurfer is equipped with rotating, tilting feet to allow four different viewing positions.



To tilt the body back slightly for bench top viewing, pull the small flaps on the bottom of the feet away from the body of the oscilloscope.

To tilt the body forward, rotate both feet to the back. This position is useful when placing the oscilloscope on a high shelf. Pulling out the flaps in this position increases the angle of the tilt.

Connecting to Other Devices/Systems

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

LAN

The oscilloscope is preset to accept DHCP network addressing over a TCPIP connection. Connect an Ethernet cable from either port on the back panel to a network access device. Go to **Utilities > Utilities Setup > Remote** to find the IP Address. See [Remote Control](#) for instructions on configuring a static IP and entering network credentials.

Go to **Utilities > Preference Setup > Email** to [configure email settings](#).

Remote Control

You can control the instrument using the IP address. Connect an Ethernet cable from either port on the back panel to a network access device, or directly to the controller. Be sure the instrument is on the same subnet as the controller.



Note: Depending on the host, you may require a cross-over cable to make a direct connection.

To switch to USB remote control, connect a USB-A/B cable from the **USBTMC port** on the back panel to a host port on the controller. Go to **Utilities > Utilities Setup > Remote** and select **Control From USBTMC**.

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 3000 supports external monitors with 1024 x 600 ppi resolution. Connect the monitor cable to the **VGA video output** on the back of the instrument. The connection is "plug-and-play" and does not require any further configuration on the oscilloscope.

Printer

WaveSurfer 3000 supports PictBridge-compliant printers. Connect the printer to any host USB port. Go to **Utilities > Utilities Setup > Hardcopy** 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** or touch the front panel **WaveSource button** to configure the signal.



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

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.

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 (V/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, simply press the front panel **Auto Setup** button.

Recall Default Setup

Restore the factory default settings by pressing the front panel **Default Setup** button. **Confirm** the action.

You can also restore default settings via the touch screen by choosing **File > Recall Setup** from the menu bar and touching the **Recall Default** button.

Using MAUI

MAUI, the Most Advanced User Interface, is Teledyne LeCroy's unique oscilloscope user interface. MAUI is designed for touch—all important controls for vertical, horizontal, and trigger are only one touch away.

Touch Screen

The touch screen is the principal viewing and control center of the oscilloscope. The entire display area is active: use your finger or a stylus to touch, touch-and-drag, or draw a selection box. Many controls that display information also work as “buttons” to access other functions.

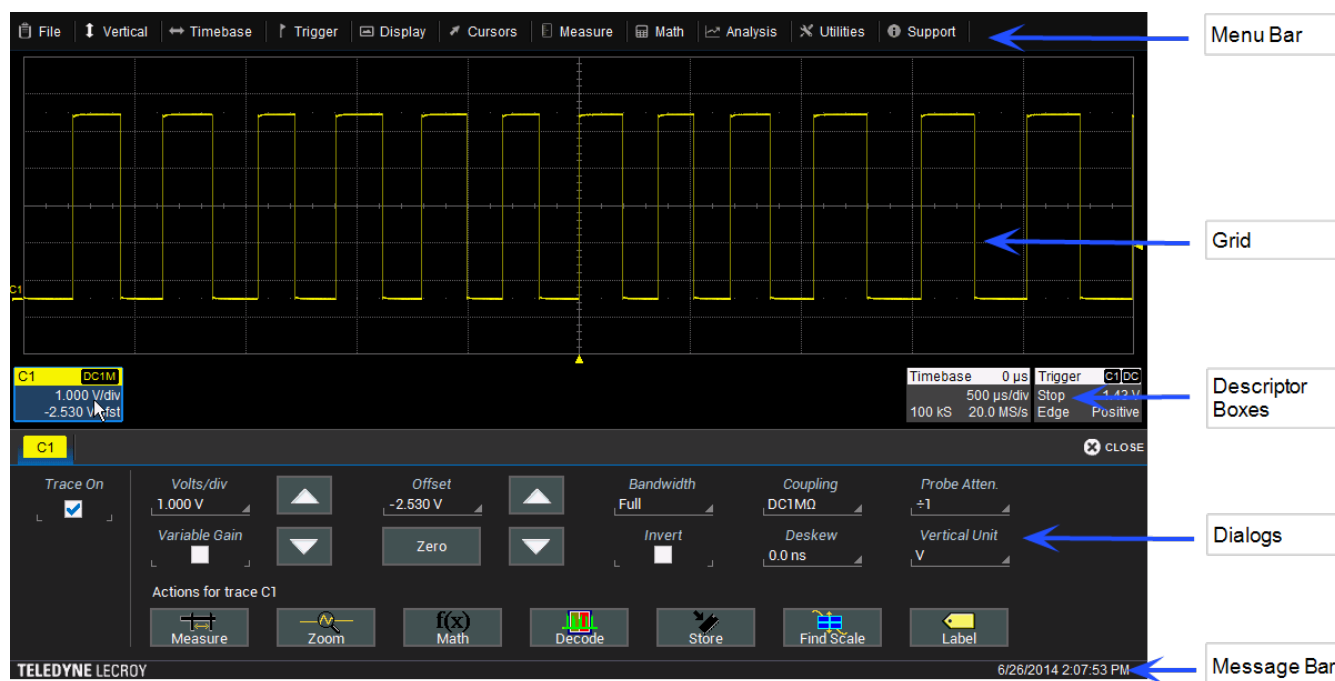
WaveSurfer3000z models feature a capacitive touch screen that supports fluid, tablet-like response to gestures.



Note: Use a capacitive stylus (not included) with these models. A regular stylus will not work.

If you have a mouse installed, you can click anywhere you can touch to activate a control; in fact, you can alternate between clicking and touching, whichever is convenient for you.

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



Menu Bar

The top of the window contains a complete menu of functions. Making a selection here changes the dialogs displayed at the bottom of the screen.

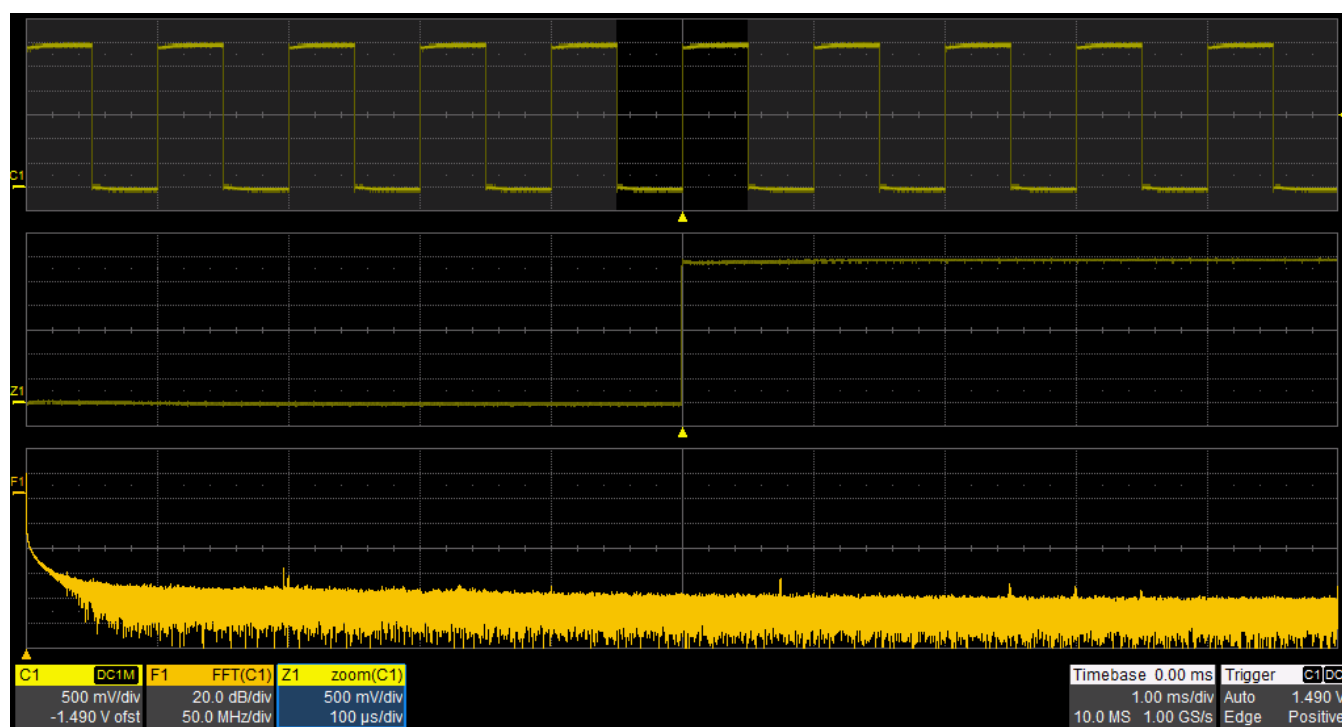
While many operations can also be performed from the front panel or launched via the descriptor boxes, the menu bar is the best way to access dialogs for Save/Recall (File) functions, Display functions, Status, LabNotebook, Pass/Fail setup, optional Analysis packages, and Utilities/Preferences setup.

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

Grid Area

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

The grid area can be divided up to three times to show channel (C_n), math (F_n), and zoom (Z_n) 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. 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.



Different types of traces opening in a separate grids.

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.



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



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



Trigger Level at the right edge of the grid tracks the trigger voltage level. If you change the trigger level when in Stop trigger mode, or in Normal or Single mode without a valid trigger, a hollow triangle of the same color appears at the new trigger level. The trigger level indicator is not shown if the triggering channel is not displayed.



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



Cursor markers appear over the grid to indicate specific voltage and time values 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, a second time to open the trace 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 (when in Sequence mode).

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

Pre-processing Symbols on Descriptor Boxes

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

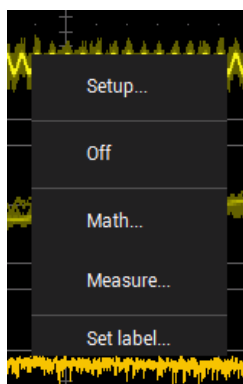
Other Trace Descriptor Boxes

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.

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		

Trace Context Menu

Touch the trace to open the trace context menu, a pop-up menu of actions to apply to the trace such as turn off, apply measurement, or label.



Timebase and Trigger Descriptor Boxes

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

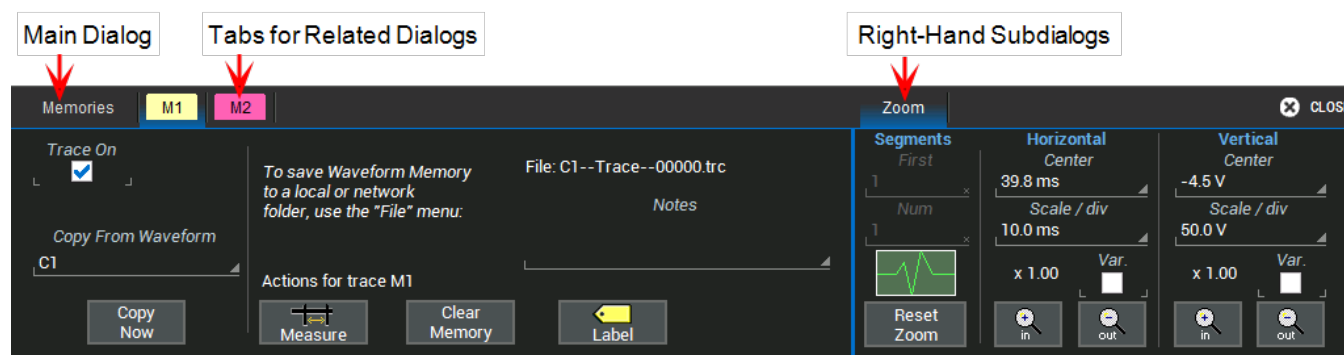
Trigger descriptor box shows: (clockwise from top right) Trigger Source and Coupling, Trigger Level (V), Slope/Polarity, Trigger Type, Trigger Mode.

Timebase	0 ns	Trigger	C1 DC
	50.0 ns/div	Stop	0.0 mV
2 kS	4 GS/s	Edge	Positive

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.

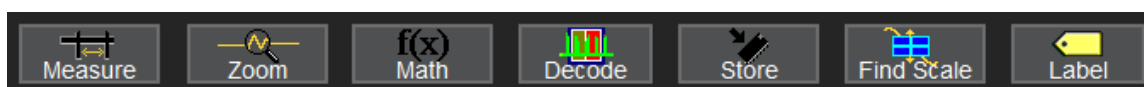


Right-Hand Subdialogs

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.

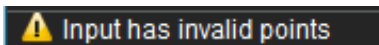
Store loads the active trace into the corresponding memory location (C1, F1 and Z1 to M1; C2, F2 and Z2 to M2, etc.).

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

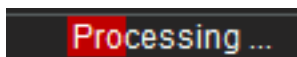
Label opens the Label pop-up to annotate the active trace.

Message Bar

At the bottom of the oscilloscope display is a narrow message bar. The current date and time are displayed at the far right. Status, error, or other messages are also 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.

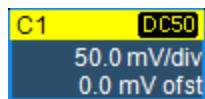


This will be especially evident when you change an acquisition setting that affects the ADC configuration while in Normal or Auto trigger mode, such as changing the Vertical Scale, Offset, or Bandwidth. Traces may briefly disappear from the display while the oscilloscope is processing.

Touch Actions

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 touch screen actions to precise values.

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.

Position Cursors

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

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 Level.

Scroll

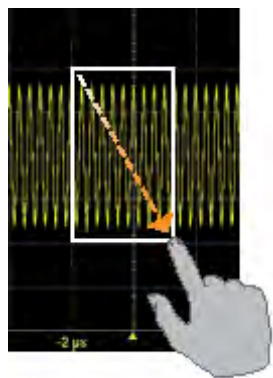
To **scroll long lists of values**, swipe the selection dialog or table in an up or down direction.

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.

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

Zoom



To **create a new zoom trace**, touch then drag diagonally to draw a selection box around the portion of the trace you want to zoom. Touch the Z_n descriptor box to open the zoom factor controls and adjust the zoom exactly. See [Zooming](#) for other ways to zoom traces.

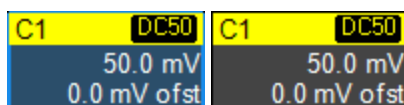
Working With Traces

Traces are the visible representations of waveforms that appear on the display grid. They may show live inputs (Cn, Digitaln), 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/reporting.

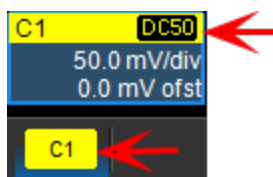
Each visible trace will have a [descriptor box](#) summarizing its principal configuration settings.

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 a 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.

Turning On/Off Traces

Turn On/Off Analog Trace

From the front panel, press the **Channel** button.

From the touch screen, choose **Vertical > Channel x Setup**.

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

- From the [Context menu](#) select **Off**.
- Clear the "Trace On" box on the *Cn* dialogs.

Turn On/Off Digital Trace

From the front panel, press the **Dig** button, then check **Group** on the *Digitaln* dialog.

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

Turn off the trace the same as you would an analog trace.

Turn On/Off Zoom Trace

See [Creating Zooms](#).

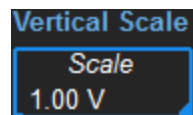
Turn On/Off Other Trace

To turn on traces, use the respective setup dialogs (Math, Memory, etc.).

Turn off the trace the same as you would an analog trace.

Adjusting Traces

To adjust Vertical Scale (gain or sensitivity) and Vertical Offset, just activate the trace and use the front panel Vertical knobs. To make other adjustments—such as channel pre-processing or the math function rescaling—touch the trace descriptor box twice to open the appropriate setup dialog.



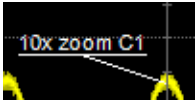
Many entries can be made by selecting from the pop-up that appears when you touch a control. When an entry field appears highlighted in blue after touching, it is *active* and can be adjusted by turning the front panel knobs. Fields that don't have a dedicated knob (as do Vertical Level and Horizontal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

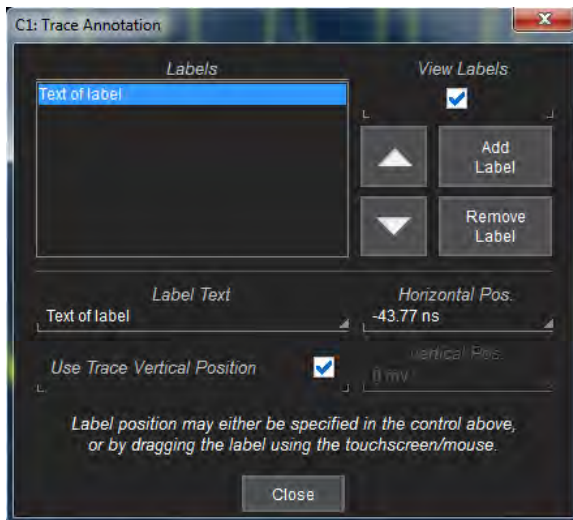
Labeling Traces

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 Scale relative to the source trace.

[Zooms may be created](#) in several ways, using either the front panel or the touch screen. You can adjust zooms the same as any other trace using the front panel Vertical and Horizontal knobs or the touch screen [zoom factor controls](#).

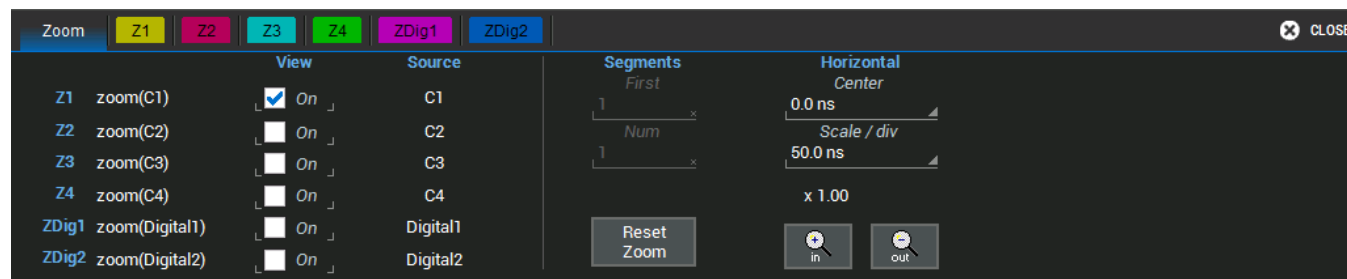
The current settings for each zoom trace can be seen on the [Zn dialogs](#).

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, such as when setting up:

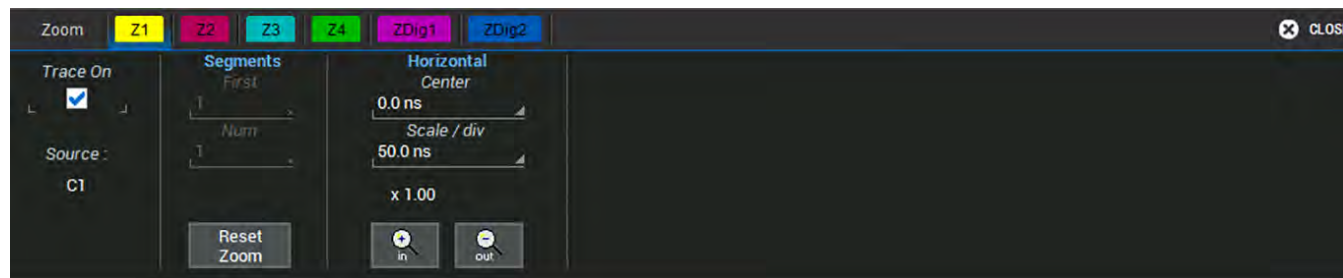
- Math traces on [Fn dialogs](#)
- Memory traces on [Mn dialogs](#)

To display the Zoom dialog, choose **Math > Zn Setup** from the menu bar, or touch any zoom descriptor box.



Zn Dialog

Each *Zn* dialog reflects the center and scale for that zoom. Use it to adjust the zoom magnification.



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 for this zoom to any channel, math, or memory trace while maintaining all other settings.

Segment Controls

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

Zoom Factor Controls

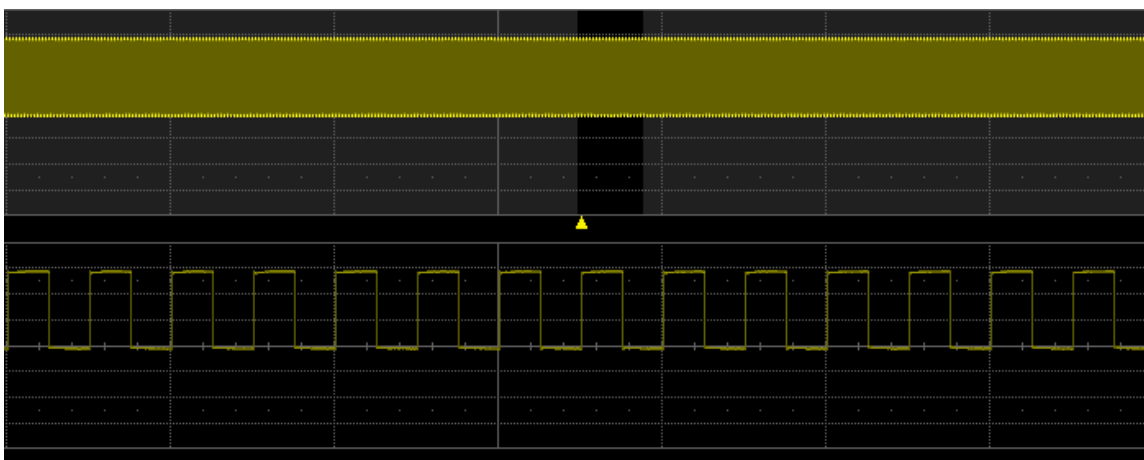
- **Out** and **In** buttons increase/decrease zoom magnification and consequently change the Horizontal Scale settings. Touch either button until you've achieved the desired level.
- **Horizontal Scale/div** sets the time represented by each horizontal division of the grid. It is the equivalent of Time/div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



Tip: On WaveSurfer oscilloscopes, all zooms are displayed in the same grid at the same horizontal scale. To use zooms to show the same source trace at different scales, always turn off one zoom before turning on another.

Creating Zooms

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



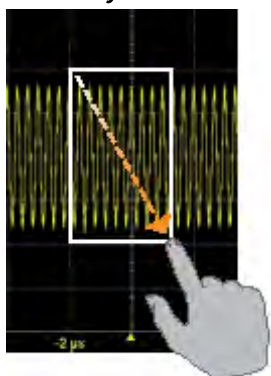
Area around zoomed section shaded.

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.

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

Manually Create Zoom



To manually create a zoom, touch-and-drag diagonally to draw a selection box around any part of the source trace.

The zoom will resize the selected area to fit the full width of the grid. The degree of vertical and horizontal magnification, therefore, depends on the size of the rectangle that you draw.

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

Adjust Zoom Scale

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

Close Zoom

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

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

Print/Screen Capture

The **front panel Print button** captures an image of the touch screen and outputs it according to your Hardcopy settings. It can be used to create an image file of waveform traces, or send the image to a networked printer or email recipient.



The **Printer** icon at the right of the Hardcopy dialog will also execute your print setting.

Print may be used as a screen capture tool by going to **Utilities > Utilities Setup > Hardcopy** and selecting to print to **File**, then choosing a graphical format. Once configured, just press the Print button or Printer icon.

You can also use the touch screen to generate a screen capture by choosing **File > Print**. The file is saved using your latest Hardcopy settings.

Acquisition

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

- [Vertical](#) axis scale at which to show the input signal and probe characteristics that affect the signal
- [Horizontal](#) axis scale at which to represent time, and acquisition 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.

Vertical

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



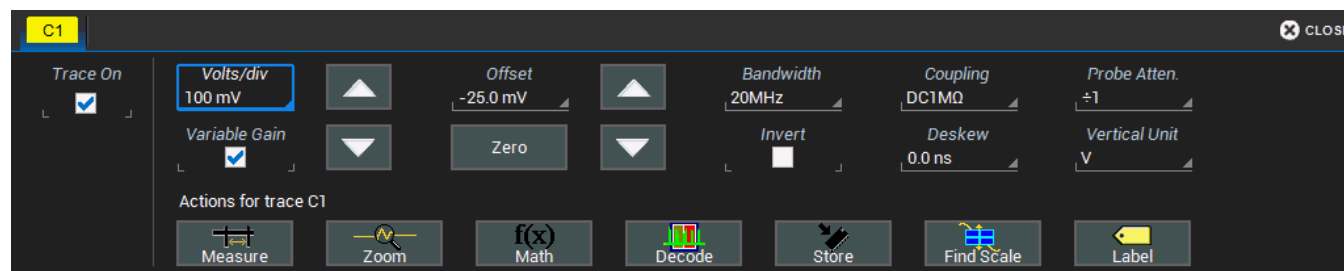
Note: While Digital settings can be accessed through the Vertical menu on Mixed Signal oscilloscopes, they are handled quite differently. See [Digital](#).

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 *Cn* descriptor box always shows the current Vertical Scale setting.

Detailed configuration for each trace is done on the [Cn dialogs](#).

Cn (Channel) Dialog

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



The *Cn* dialog contains:

- Vertical settings for scale, offset, coupling, bandwidth
- Units settings
- Probe attenuation and deskew

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

Vertical Units may be changed from Volts (V) to Amperes (A). This is useful when using a third-party current probe (which is not auto-detected) or when probing across a current sensor/resistor.

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.

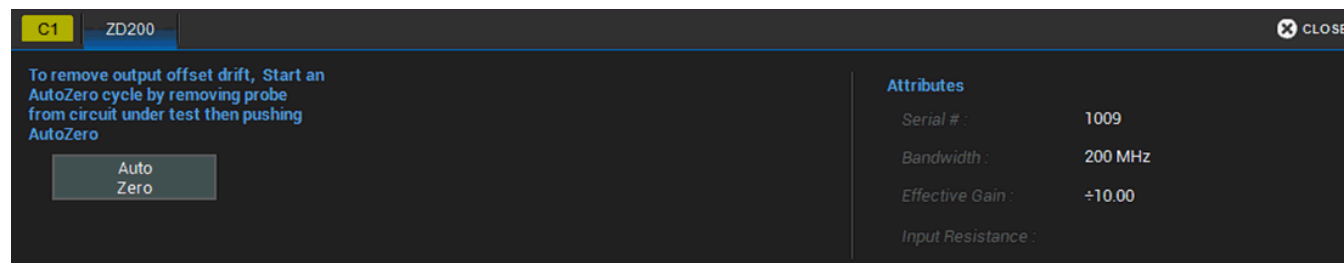
Deskew adjusts the horizontal time offset by the amount entered in order to compensate for propagation delays caused by different probes or cable lengths. The valid range is dependent on the current timebase setting. The Deskew pre-processing setting and the Deskew math function perform the same action.

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 AutoZero, DeGauss and make tip selection for probes from the touch screen. Other settings may appear, as well, depending on the probe model.



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



Auto Zero Probe

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.

DeGauss Probe

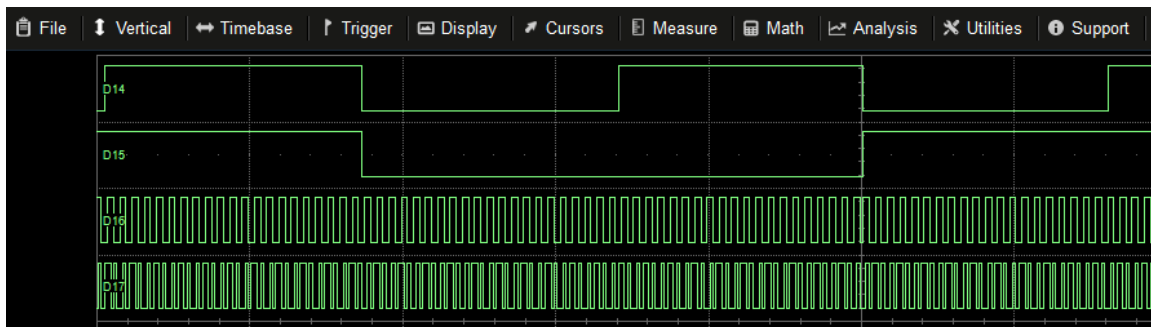
The Degauss control is activated for some types of probes (e.g., current probes). Degaussing eliminates residual magnetization from the probe core caused by external magnetic fields or by excessive input. It is recommended to always Degauss probes prior to taking a measurement.

Digital (Mixed Signal)

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

Digital Traces

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



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

Activity Indicators

Activity indicators appear at the bottom of the Digital n dialogs when using the Digital Leadset. 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 and renamed appropriately depending on the group. 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 numbered the same as the physical lead they represent, although any line number can be re-assigned to any lead on the Digital setup dialogs.

Connecting/Disconnecting the Leadset

The digital leadset connects to the Mixed Signal interface on the front of the instrument.

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.



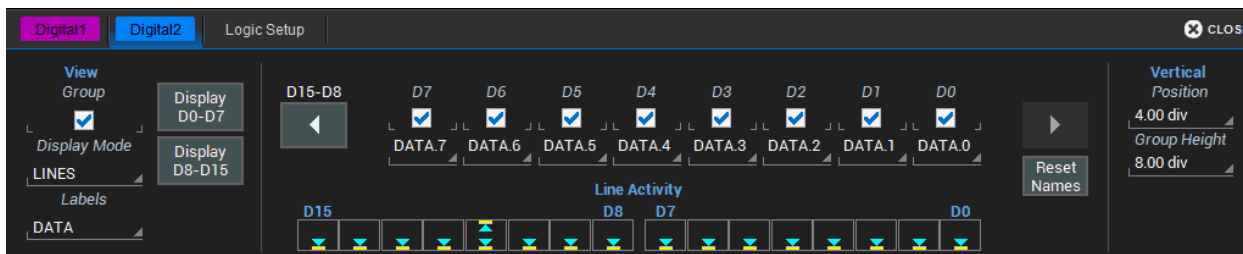
Each flying lead has a signal and a ground connection. A variety of ground extenders and flying ground leads are available for different probing needs.

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. Connect the digital input device to the test device and the instrument.
2. From the menu bar, choose **Vertical > Digital n Setup**, or press the front panel **Dig** button and select the desired **Digital n** tab.

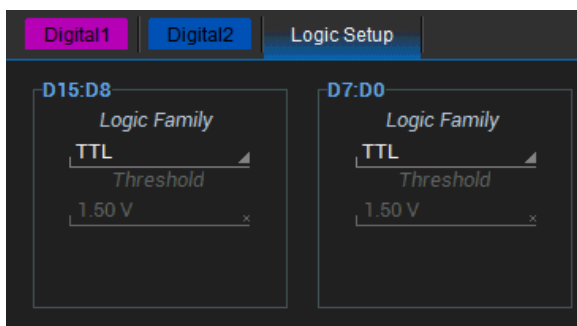


3. On the Digital n set up dialog, check the boxes for all the lines that comprise the group. Touch the Right and Left Arrow buttons to switch between digital banks as you make line selections.



Note: Each group can consist of anywhere from one to all of the leads from any digital bank. It does not matter if the some or all of the lines have been included in other groups.

4. Check **View Group** to start the display.
5. When you're finished on the Digital n dialog, open **Logic Setup** and choose the **Logic Family** that applies to each digital bank, or set custom **Threshold** values.



Digital Display Setup

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

1. Choose a **Display Mode**:

- **Lines** (default) shows a time-correlated trace indicating high, low, and transitioning points (relative to the Threshold) for every digital line in the group. The size and placement of the lines depend on the number of lines, the Vertical Position and Group Height settings.
- **Bus** collapses the lines in a group into their Hex values. It appears immediately below all the Line traces when both are selected.
- **Line & Bus** displays both types of digital trace.

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

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

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

Renaming Digital Lines

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

1. Touch **Label** and select from:
 - **Data** - the default, which 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.
2. 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**.

Also, labels can 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.

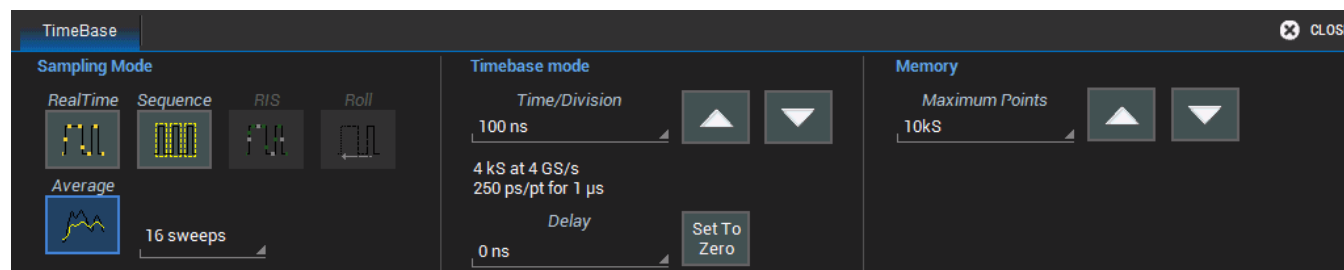
Timebase

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, memory /sample rate, and number of active channels. You can also use it instead of the Front Panel to modify the Time/Div and horizontal Delay. There are related dialogs for [Reference Clock](#).



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.



Note: RIS and Roll modes are not available when using any other type of digital interleaving.

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 current Sample Rate and Time/Division 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.

To avoid aliasing and other waveform distortions, it is advisable (per Nyquist) to acquire at a sample rate at least twice the bandwidth of the input signal. Use Maximum Points in relation to Time/Division to adjust the overall Sample Rate (shown on the Timebase descriptor). The formula for sample rate is: *Sample Rate* = *Memory Samples/Acquisition Time*, with the maximum sample rate being limited by the instrument's analog-to-digital converter (ADC).

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.

RIS Sampling Mode

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

The maximum effective RIS sampling rate is achieved by making multiple single-shot acquisitions at maximum real-time sample rate. The bins thus acquired are positioned approximately 20 ps (50 GS/s) apart. The process of acquiring these bins and satisfying the time constraint is a random one. The relative time between ADC sampling instants and the event trigger provides the necessary variation.

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



Note: RIS is not available when the oscilloscope is using another form of interleaving.

Roll Sampling Mode

Roll mode displays, in real time, incoming points in single-shot acquisitions that appear to "roll" continuously across the screen from right to left until a trigger event is detected and the acquisition is complete. The parameters or math functions set on each channel 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 and decrease sample rate to 2.5 MS/s or less to activate the Roll mode option on the Timebase dialog. Only Edge trigger is supported for Roll mode acquisitions.



Note: Roll sampling is not available when using 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 waveform consists of a number of fixed-size segments. The instrument uses the Timebase Sequence settings to determine the capture duration of each segment. The desired number of segments, maximum segment length, and total available memory are used to determine the actual number of samples or segments, and time or points.

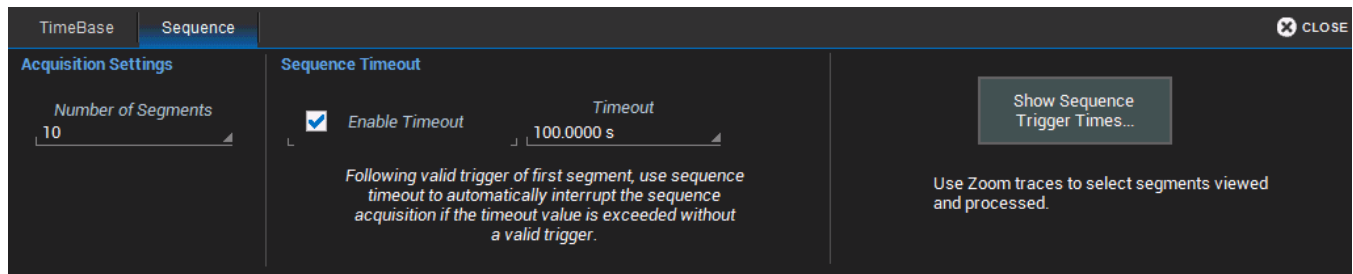
Sequence Mode is ideal when capturing many fast pulses in quick succession or when capturing few events separated by long time periods. The instrument can capture complicated sequences of events over large time intervals in fine detail, while ignoring the uninteresting periods between the events. Measurements can be made on selected segments using the full precision of the timebase.



Note: You cannot operate Serial Decoders on Sequence Mode acquisitions.

Sequence Mode Set Up

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



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



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

4. Touch one of the **front panel Trigger buttons** to begin acquisition.

You can interrupt acquisition at any time by pressing the front panel **Stop** button. In this case, the segments already acquired will be retained in memory.

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 value to 1, you can use the front panel Adjust knob to scroll through each segment in order.

Channel descriptor boxes indicate the total number of segments acquired in sequence mode. Zoom descriptor boxes show the segment currently displayed. As with all other zoom traces, the zoomed segments are highlighted on the source trace.

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**.

1. Touch **First Selected** and choose the first segment to display.
2. Touch **Number of Selected** and enter the total number of segments to display.

Set Reference Clock

By default, the oscilloscope is set to use its internal clock of 10 MHz as the Timebase reference to synchronize acquisition across all channels.

You can opt to use an external reference clock for this purpose. Connect the clock source to the **REF IN** input on the back I/O panel of the oscilloscope using a BNC cable. Then, go to **Timebase > Timebase Setup > Reference Clock tab** and choose **External**.

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 individual details or changes in the waveforms over time.

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.

Press the front panel **History Mode** button, or choose **Timebase > History Mode** to access this feature.

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



Note: History Mode does not work with Sequence Mode acquisitions, pre-processor Interpolation set on the input channel, or 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 Navigation buttons to "scroll" the history of acquisitions.

- The top row buttons scroll: Fast Backward, Slow Backward, Slow Forward, Fast Forward.
- The 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. View individual acquisitions by selecting the row from the table or entering 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, many triggers can be set on either analog channels, including the External Trigger input, or digital lines.

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 MultiStage 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. Special MultiStage types allow one set of conditions for arming and another (same or different) set for triggering.

If a type is part of a subgroup (e.g., Smart), first choose the subgroup to display all the trigger options.

Basic Types

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

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

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

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

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 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.



Note: This functionality is identical to Teledyne LeCroy's previous Qualify and State triggers, but presented through a different user interface.

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.

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

Interval triggers upon finding a specific interval, the time (period) between two consecutive edges of the same polarity: positive to positive or negative to negative. Use the interval trigger 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.

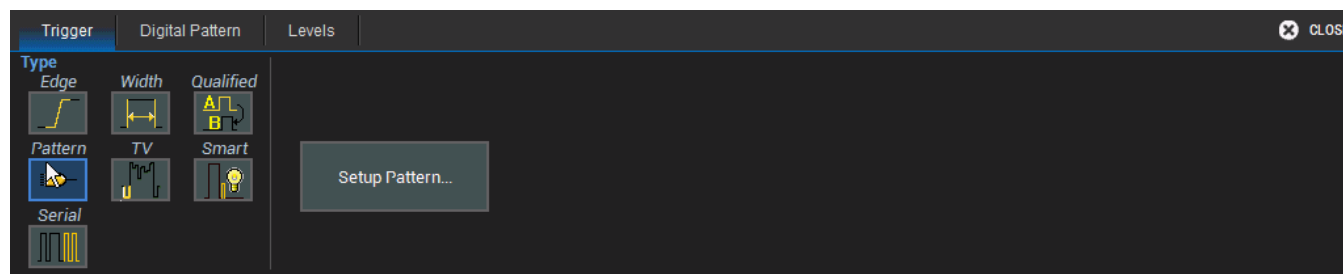
SlewRate triggers when the rising or falling edge of a pulse crosses an upper and a lower level. The pulse edge must cross the thresholds faster or slower than a selected period of time.

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).

Coupling

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

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

Slope/Polarity

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

For others, such as Width, the equivalent selection will be **Polarity** (positive vs. negative).

Level

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

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

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

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. The values may be **In Range** that is bounded by the upper/lower values or **Out Range**.

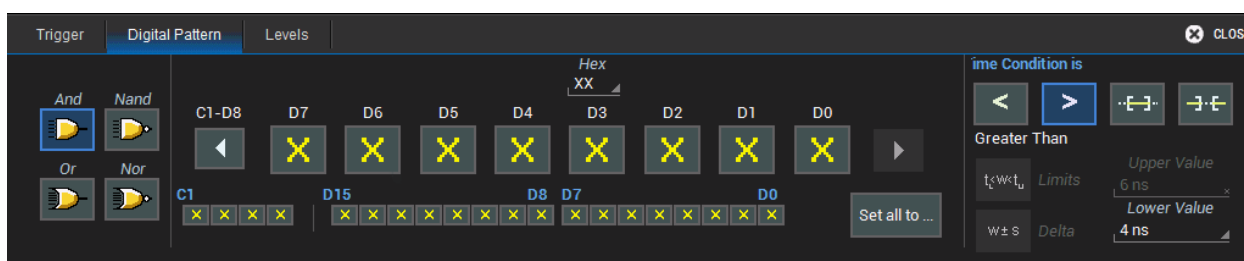
The extent of the range can often also be specified by using a **Nominal** and **Delta** value, rather than an absolute upper and lower value. In this case, the Nominal value sets the center of the range, and the Delta determines how many units plus/minus the Nominal value are included in the range.

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 pattern of low/high states across multiple inputs is used by the Pattern type. Patterns 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 **On** button for each active line, and select whether it must be High or Low compared to the logic threshold. Depending on your selection, 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.

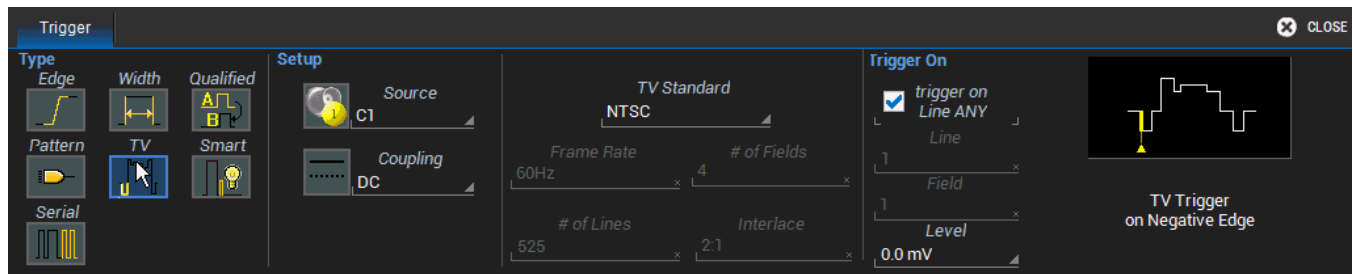


Note: Analog patterns always assume a logical "And" when combined with any digital pattern. Both conditions must be true for the trigger to fire.

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 (PAL, SECAM, NTSC, HDTV) or custom composite video signals.



1. Choose the **Source** signal input.
2. Choose the signal **TV Standard**. To use a custom signal, also enter the **Frame Rate**, **# of Fields** per line, **# of Lines**, and **Interlace** ratio.
3. Choose the **Line** and **Field** upon which to trigger.

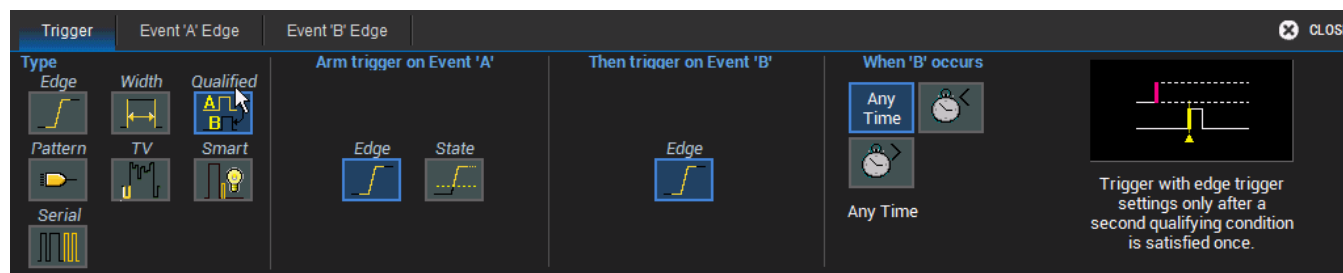
Serial Trigger

The Serial trigger type will appear if you have installed serial data trigger and decode options. Select the **Serial** type then the desired **Protocol** to open the serial trigger setup dialogs. For setup instructions, see the software instruction manual at [_____](#) under Manuals > Software Options.

Qualified Trigger

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

On the Trigger dialog, select **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.

When B is an Edge or Pattern, 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 triggers. Holdoff disables the trigger temporarily, even if the other conditions are met. Use Holdoff to obtain a stable trigger for repetitive, composite waveforms. For example, if the number or duration of sub-signals is known, you can disable them by setting an appropriate Holdoff value.

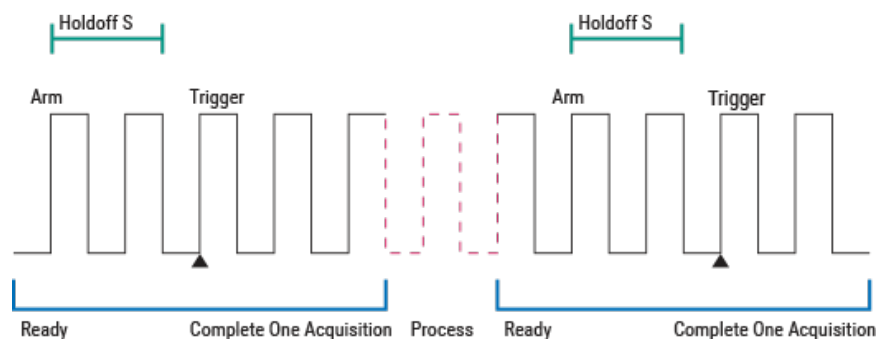


Note: [Qualified](#) triggers operate using time or event conditions similar to Holdoff, but arm and trigger differently.

Hold Off by Time

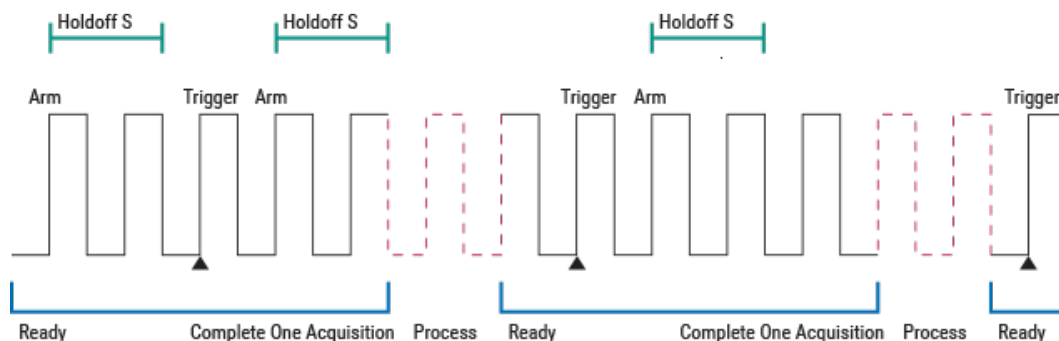
This is a period of time to wait after the arming event before triggering on the next event. The maximum allowed time is 20 seconds; the Holdoff time would otherwise be limited only by the input signal, the coupling, and the instrument's bandwidth.

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



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

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



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



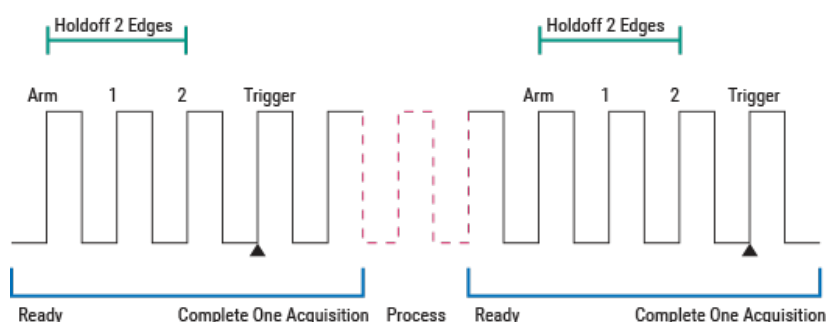
Note: Because there is only one trigger per acquisition, the trigger event will always belong to the new acquisition. The processing time shown here is for purposes of illustration only.

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

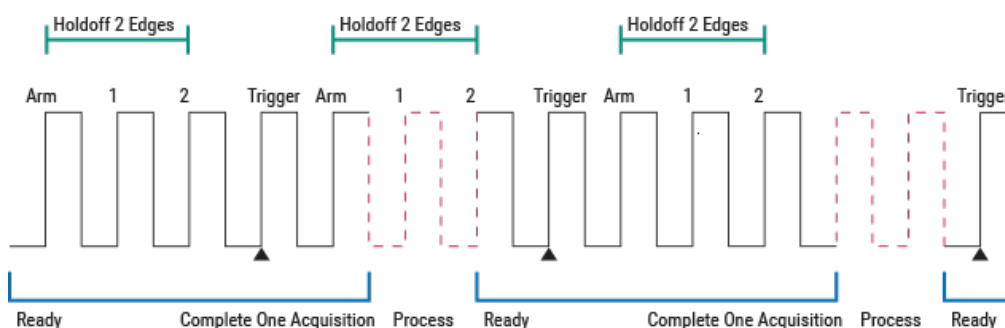
Events refers to the number of times the trigger conditions have been met following the arming event.

For example, if the Holdoff is two edges counted from the start of the acquisition, the oscilloscope reads, 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, the Holdoff appears to be "Holdoff plus one."



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

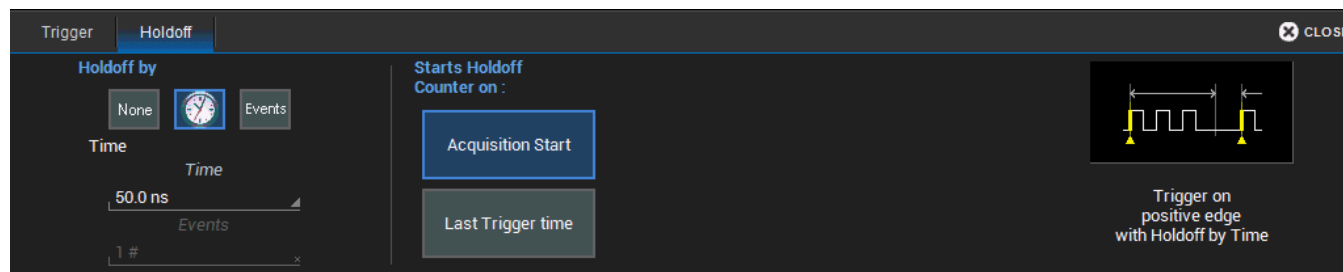
As with Holdoff by time, when a Holdoff by events is counted from the last trigger time, the oscilloscope re-arms immediately following the trigger and begins the Holdoff count. If the count is satisfied by the time the oscilloscope is again ready, the trigger occurs on the next event at the start of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



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

Holdoff Set Up

To add Holdoff to an Edge 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.

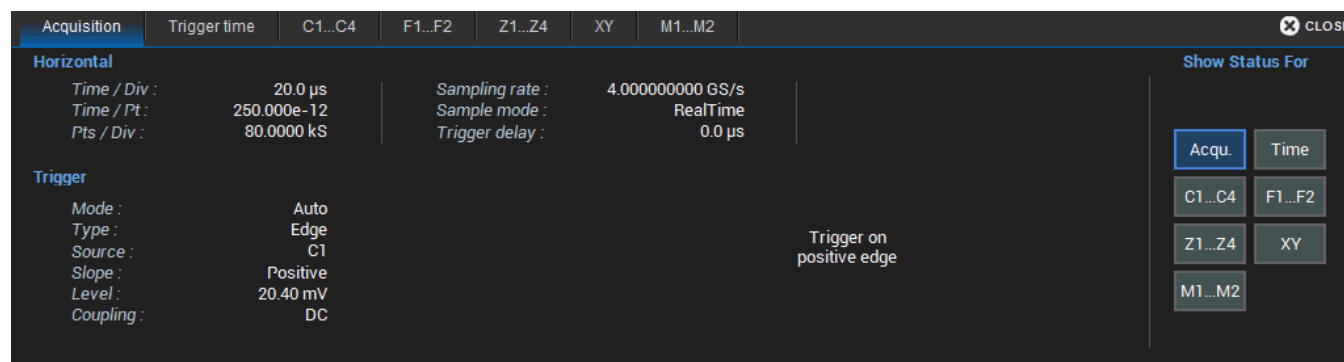
Choose to **Start Holdoff Counter On:**

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

Viewing Status

All instrument settings can be viewed through the various Status dialogs. These show all existing acquisition, trigger, channel, math function, measurement and parameter configurations, as well as which are currently active.

Access the Status dialogs by choosing the Status option from the Vertical, Timebase, Trigger, Math, or Analysis menus (e.g., Channel Status, Acquisition Status). The available options will depend on your model.



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.

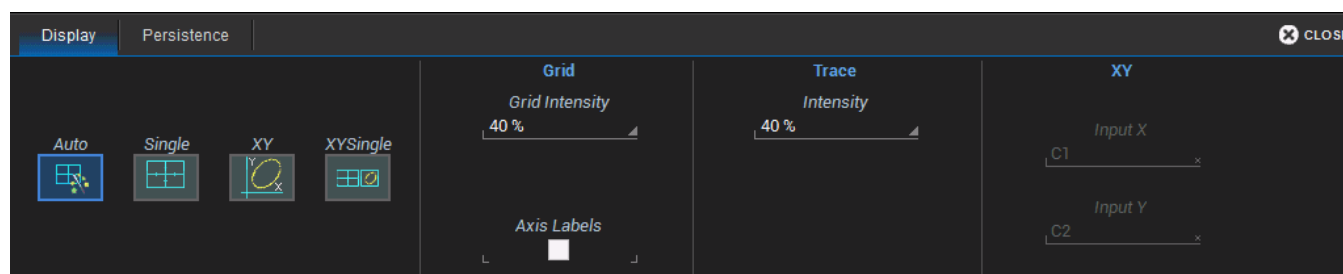
Auto Grid is enabled by default. This feature divides the screen as needed when new traces open. WaveSurfer oscilloscopes may be divided into a maximum of three grids—one each for channels/memories, math functions, and zooms—that each represent the full number of vertical levels. All traces of the same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

Grid on top superimposes the grid over the waveform.



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

On WaveSurfer oscilloscopes, **Axis labels** display the values associated with the top and bottom grid lines (calculated from Volts/div) and the time associated with the extreme left and right grid lines (calculated from the Time/div).

Trace Intensity

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

When more data is available than can actually be displayed, 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**.

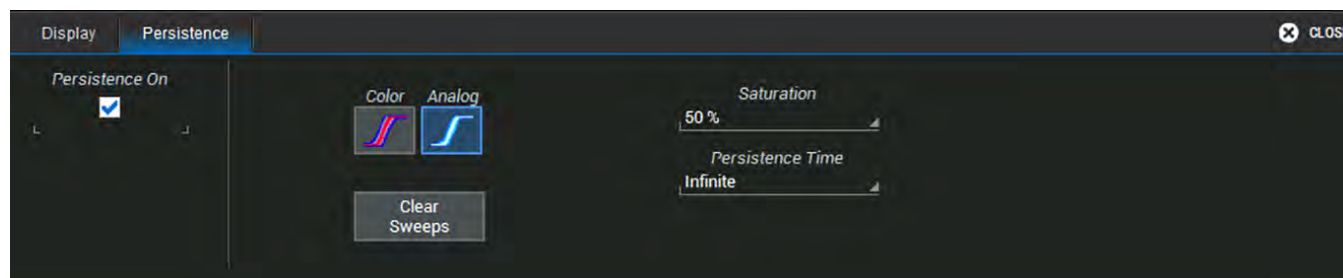
Persistence Display

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.

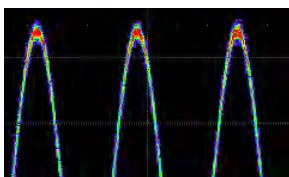
The different persistence modes show the most frequent signal path in three-dimensional intensities of the same color (Analog), or in a graded spectrum of colors (Color).

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

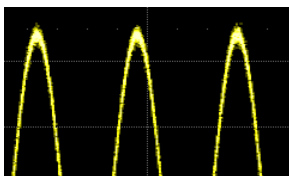


Apply Persistence

1. Check **Persistence On**.
2. Use the buttons to select a persistence mode:



Color Mode persistence works on the same principle as Analog persistence, but instead uses the entire color spectrum to map signal intensity: 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.



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. The information in the lower populations (for example, down at the noise level) could be of greater interest to you than the rest. The Analog persistence view highlights the distribution of data so that you can examine it in detail.

3. Select the **Saturation** level as a percentage of the maximum population. All populations above the saturation population are assigned the highest color intensity: that is, they are saturated. 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 the percentage value specified. Lowering this percentage causes the pixels to be saturated at a lower population and makes 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.

Remove Persistence

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

Math and Measure

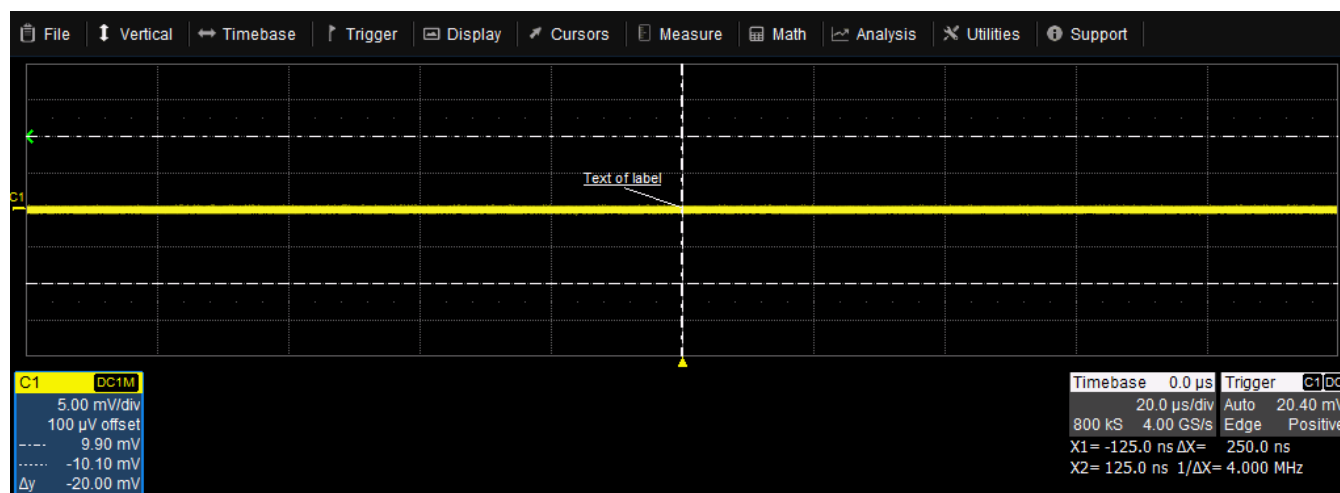
Teledyne LeCroy offers a rich set of standard, pre-programmed tools for the "quickest time to insight" into the characteristics of acquired waveforms. Most instruments calculate measurements for all samples in an acquisition, enabling you to rapidly and thoroughly calculate thousands or millions of parameter values and apply a variety of mathematical functions to the input waveform trace.

Cursors

Cursors are markers (lines or cross-hairs) that identify voltage and time values on the waveform. Use cursors to make fast, accurate measurements of specific points in the waveform. There are three, standard [cursor types](#) available.

Vertical (amplitude) cursor readouts appear the trace descriptor box; Horizontal (time) cursor readouts appear below the Timebase descriptor box.

Cursors can be placed on math functions whose X-axis has a dimension other than time, such as an FFT.



Horizontal cursors.

Cursor Types

These cursors can be placed on most Channel, Memory, Math or Zoom traces:

- **Horizontal (Time)** cursors intersect two points on the horizontal axis. The readout shows absolute values and a delta of the two points.
- **Vertical (Amplitude)** cursors intersect two points on the vertical axis. The readout shows absolute values and a delta of the two points.
- The **Horizontal + Vertical** option places both cursors together.

Two other cursors are offered only in special circumstances:

- **Horizontal (Frequency)** cursors look the same as Horizontal (Time) cursors except that they are placed on waveforms that have frequency on the x-axis, such as FFTs.
- **Horizontal (Event)** cursors are placed only on Trend waveforms.

Apply and Position Cursors

Apply 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 waveform on the grid for cursors to execute, although acquisition may be in process or stopped when you turn them on.

To turn off cursors, choose **Cursors > Off** from the menu bar, or continue cycling the Cursor button until you reach "Off" (the cursor lines disappear).

Position Cursors

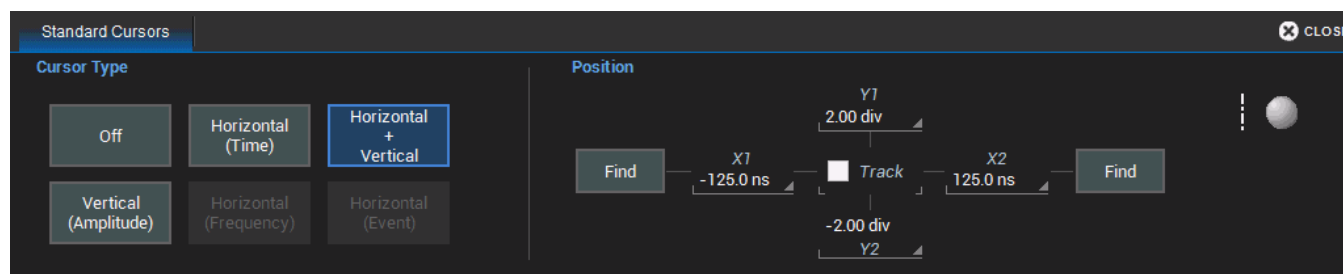
The easiest way to reposition a cursor is to drag-and-drop the cursor marker to a new position. Indicators outside the grid show to which trace the cursor belongs when you have multiple traces on one grid.

Alternatively, use the front panel **Cursor knob**. Push the knob until the correct line is selected, then turn to move it. The third press of the Cursor knob selects both cursors so they will track together when the knob is turned.

When there are multiple traces on the same grid, 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.

Standard Cursors Dialog

These controls can be used instead of the front panel controls to set 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.

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. The options available depend on the Cursor Type settings.

- **X 1** (negative) and **X 2** (positive) sets time from the zero point.
- **Y 1** (negative) and **Y 2** (positive) sets number of divisions from the zero level. It may be entered as time or a fraction of a division.
- **Track** locks cursor lines so they move together, maintaining the same distance from each other.
- **Find** places the cursor 2.5 divisions (negative or 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 the cursors on the XY trace, the same as you use the settings on the [Standard Cursors](#) dialog for VT traces. The functionality is the same.

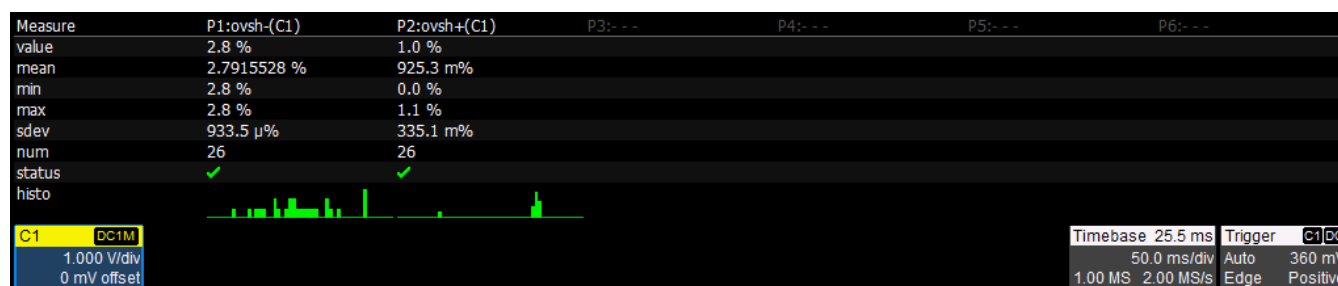
Measure

Measurement parameters are tools that give you access to a wide range of waveform properties. Use them to analyze many attributes of your waveform such as rise-time, rms voltage, and peak-to-peak voltage. Measurements can also be graphed as a trend for statistical analysis.

Create a custom set of parameters drawn from all available [measurements](#).

Measure Table

Measurement readouts appear in a table below the grid. The **value** row shows the measurements taken for each parameter on the last cycle of the last acquisition. Other rows show optional statistics.



Measurement Status Indicators

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

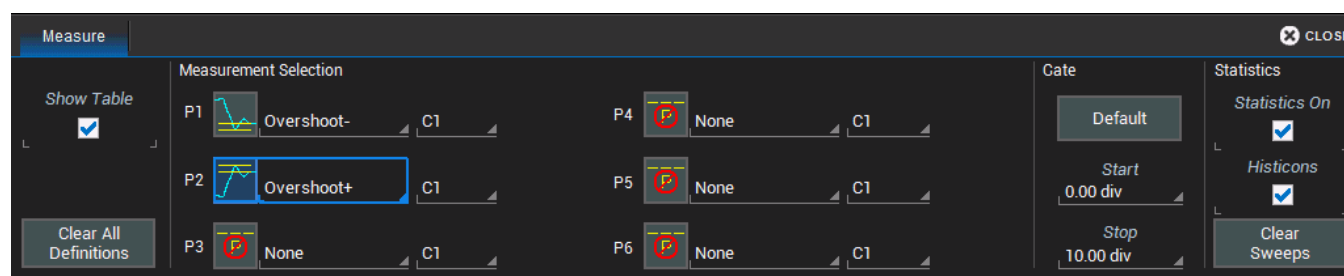
Symbol	Description
	Problem with the signal or the setup. Touch cell to see explanation in message bar.
	Valid value returned.
	Unable to determine top and base; however, measurement may still be valid.
	Underflow condition.
	Overflow condition.
	Simultaneous underflow and overflow condition.

Navigating with the Table

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

Parameter Set Up

The Measure Dialog gives quick access to measurement features. Besides configuring parameters, use the Measure dialog to show [Statistics](#) and [Histicons](#), or to [Gate](#) measurements.



1. To open the Measure dialog, choose **Measure > Measure Setup** from the menu bar.
2. Check **Show Table** to display the measurement readout on screen.
3. For each parameter (**P_n**):
 - Touch the **Measurement** 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.

Clear All Definitions resets all parameters to "None". Selections cannot be restored after clearing, you must repeat parameter set up.

Statistics

Checking **Statistics On** adds the measures mean, min., max., sdev, and num(ber of measurements computed) to the table.

For any parameter that computes on an entire waveform (like amplitude, mean, minimum, maximum, etc.), the num statistic represents the number of sweeps.

For any parameter that computes on every event, the num statistic represents the number of events per acquired waveform. If x waveforms were acquired, num is x times the number of cycles per waveform.

To reset the statistics counter, touch **Clear Sweeps** on the display or front panel.

Histicons

Histicons are miniature histograms of measurement parameters that appear on the measurement table. These thumbnail histograms let you see at a glance the statistical distribution of each parameter. Select the **Histicons** checkbox to turn on histicons.

Gating Measurements

By using gates, you can narrow the span of the waveform on which to perform tests and measurements, allowing you to focus on the area of greatest interest. For example, if you "gate" five rising edges of the waveform, rise time calculations are performed only on the five pulses bounded by the gate posts.

The default starting positions of the gate posts are 0 div and 10 div, which coincide with the left and right ends of the grid. Therefore, the gate initially encloses the entire waveform.

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 setting down to hundredths of a division by using the **Gate Start** and **Stop** fields. All parameters share the same gate.

Touch the **Default** button to return gates to the width of the trace.

Plotting Measurements

Both Track and Trend are tools that can be used to plot measurement data and observe variations with respect to time. The table below summarizes their differences.

	Track	Trend
Represents	Parameter value vs. time	Parameter value vs. event
Cumulative	Non-cumulative (resets after every acquisition), unlimited number of events	Cumulative over several acquisitions up to one million events
Time correlates to other data	Yes	No
Monitors Frequency domain	Yes	No, points are not evenly spaced in time and therefore cannot be used for an FFT
Monitors parameters over multiple acquisitions	No, resets after every acquisition	Yes
Preserves all measurement data	Yes (although maximum time period that can be captured is limited by acquisition memory and sampling rate)	No, data can be missed during time oscilloscope takes to re-arm between acquisitions

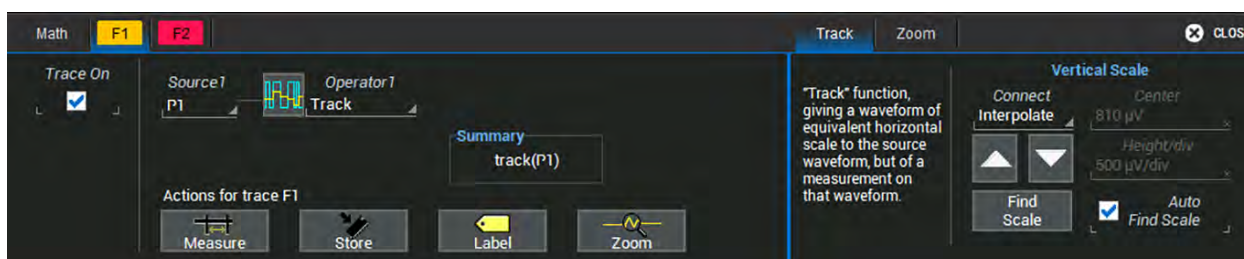
Plot Track

A Track is a plot composed of parameter measurements that is time synchronous with the source waveform. The vertical units are those of the source parameter and the horizontal units are seconds. In order to maintain time synchronism, the parameter values are posted at the sampling rate.

In general, Track is the tool to use if you want to capture a continuous stream of data spaced closely together. Track values are redundant in that the same value is repeated every sample period until the measurement changes.

Although a Track plots measurement values, it is created as a Math function and controlled on the Function dialogs.

1. On the *F_n* dialog, choose the **Track Operator**.



2. On the Track subdialog, use **Find Scale** to automatically find suitable values, or uncheck **Auto Find Scale** and enter a custom **Center** and **Height/div**.
3. Choose a line **Connect** function of either Interpolate or Extend.

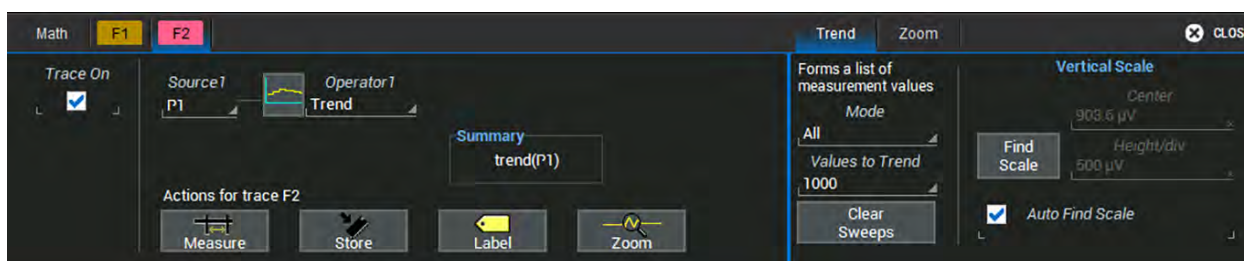
Plot Trend

A Trend is a plot composed of a series of parameter measurements 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 measurement. Think of Trend as a strip chart recorder for your instrument.

Trends are especially useful for visualizing the history of a parameter over an extended period of time or over multiple acquisitions. Trend can be used if your data is spaced widely apart and longer than the dead-time between acquisitions.

Although a Trend plots measurement values, it is created as a Math function and controlled on the Function (*F_n*) dialogs.

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



2. Choose a computation **Mode** of All (plots multiple points per acquisition) or Average (plots one point per acquisition). Enter the number of measured **Values to Trend**.
3. To rescale the Trend, use **Find Scale** to automatically find suitable values, or uncheck **Auto Find Scale** and enter custom **Center** and **Height/div** values.

List of Standard Measurements

Measurements included standard with the oscilloscope are listed below alphabetically.



Note: There may be additional parameters available depending on the software options installed on the oscilloscope.

Measurement	Description
Amplitude (ampl)	Measures the difference between upper and lower levels in two-level signals. Differs from pkpk in that noise, overshoot, undershoot, and ringing do not affect the measurement. Amplitude is calculated by using the formula $\text{Top} - \text{Base}$. On signals not having two major levels (such as triangle or saw-tooth waves), the amplitude parameter returns the same value as peak-to-peak.
Area	Integral of data: Computes area of the waveform relative to zero level. Values greater than zero contribute positively to the area; values less than zero, negatively.
Base	Lower of two most probable states (higher is top). Measures lower level in two-level signals. Differs from min in that noise, overshoot, undershoot, and ringing do not affect measurement. On signals not having two major levels (such as triangle or saw-tooth waves), the amplitude parameter returns the same value as minimum.
Delay	Time from trigger to transition: Measures time between trigger and first 50% crossing of specifies signal. Delay can be used to measure the propagation delay between two signals by triggering on one and determining delay of other.
Duty Cycle	Percent of period for which data are above or below the 50% level of the signal.
Fall 80-20% (fall8020)	Duration of pulse waveform's falling transition from 80% to 20% of the amplitude averaged for all falling transitions between the measurement gates. On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving less predictable results.
Fall time (fall)	Duration of pulse waveform's falling transition from 90% to 10% of the amplitude averaged for all falling transitions between the measurement gates. On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving less predictable results.
Frequency (freq)	Period of cyclic signal measured as time between every other pair of 50% crossings. Starting with first transition after left measurement gate. The period is measured for each transition pair. The reciprocal of each period measurement is calculated as the frequency.
Maximum (max)	Measures highest point in waveform. Unlike top, does not assume waveform has two levels.
Mean	Average of data for time domain waveform. Computed as centroid of distribution for a histogram of the data values.
Minimum (min)	Measures the lowest point in a waveform. Unlike base, does not assume waveform has two levels.
None	Disables parameter calculation.
Overshoot-	Amount of overshoot following a falling edge. This is represented as percentage of amplitude. Overshoot- is calculated using the formula $(\text{base} - \text{min.})/\text{ampl} \times 100$. On signals not having two major levels (triangle or saw-tooth waves, for example), may not give predictable results.
Overshoot+	Amount of overshoot following a rising edge specified This is represented as a percentage of amplitude. Overshoot+ is calculated using the formula $(\text{max.} - \text{top})/\text{ampl} \times 100$. On signals not having two major levels (triangle or saw-tooth waves, for example), may not give predictable results.

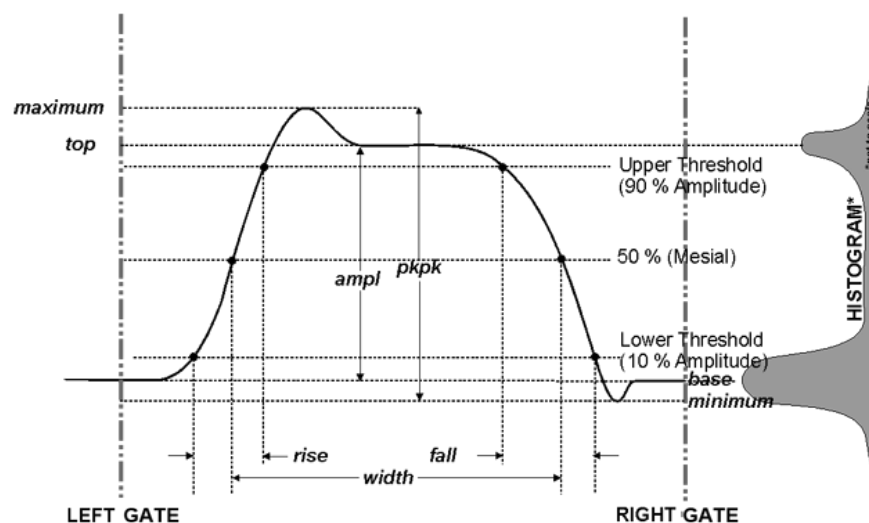
Measurement	Description
Peak to Peak (pkpk)	Difference between highest and lowest points in waveform. Unlike ampl, does not assume the waveform has two levels. Peak to peak is calculated using the formula <i>maximum – minimum</i> .
Period	The time between every other pair of 50% crossings. Starting with first transition after left measurement gate, period is measured for each transition pair, with values averaged to give final result.
Phase	Phase difference between signal analyzed and signal used as reference. Both signals are measured from the 50% point of their rising edges.
Rise 20-80% (rise2080)	Duration of pulse waveform's rising transition from 20% to 80% of the amplitude averaged for all rising transitions between the measurement gates. On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving less predictable results.
Rise Time	Duration of pulse waveform's rising transition from 10% to 90% of the amplitude averaged for all rising transitions between the measurement gates. On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving less predictable results.
RMS	Root Mean Square of data between the measure gates calculated using the formula: $\sqrt{\frac{1}{N} \sum_{i=1}^N (v_i)^2}$ Where: vi denotes measured sample values, and N = number of data points within the periods found up to maximum of 100 periods.
Skew	Time of clock1 edge minus time of nearest clock2 edge. Both signals are measured from the 50% point of their rising edges.
Std Dev (sdev)	Standard deviation of the data between the measure gates using the formula: $\sqrt{\frac{1}{N} \sum_{i=1}^N (v_i - \text{mean})^2}$ Where: vi denotes measured sample values, and N = number of data points within the periods found up to maximum of 100 periods. This is equivalent to the rms for a zero-mean waveform. Also referred to as AC RMS
Top	Higher of two most probable states (base is lower). Measures higher level in two-level signals. Differs from max in that noise, overshoot, undershoot, and ringing do not affect measurement. On signals not having two major levels (such as triangle or saw-tooth waves), the amplitude parameter returns the same value as minimum.
Width	Width of cyclic signal determined by examining 50% crossings in data input. If first transition after left cursor is a rising edge, waveform is considered to consist of positive pulses and width the time between adjacent rising and falling edges. Conversely, if falling edge, pulses are considered negative and width the time between adjacent falling and rising edges. For both cases, widths of all waveform pulses are averaged for the final result.
WidthN (widn)	Time of cyclic signal determined by examining 50% crossings in data input. The widthN is measured from falling edge to rising edge.

Calculating Measurements

The instrument uses the following methods to calculate measurements.

Determining Top and Base Lines

Proper determination of the top and base reference lines is fundamental for ensuring correct parameter calculations. The analysis begins by computing a histogram of the waveform data over the time interval spanned by the left and right measurement gates. For example, the histogram of a waveform transitioning in two states will contain two peaks (see figure). The analysis will attempt to identify the two clusters that contain the largest data density. Then the most probable state (centroids) associated with these two clusters will be computed to determine the top and base reference levels: the top line corresponds to the top and the base line to the bottom centroid.



Determining Rise and Fall Times

Once top and base are estimated, rise and fall times are calculated. The appropriate threshold levels are automatically determined by the instrument, using the amplitude (ampl) parameter.

Rising Edge Duration

$$\frac{1}{M_r} \sum_{i=1}^{M_r} (Tr_i^{90} - Tr_i^{10})$$

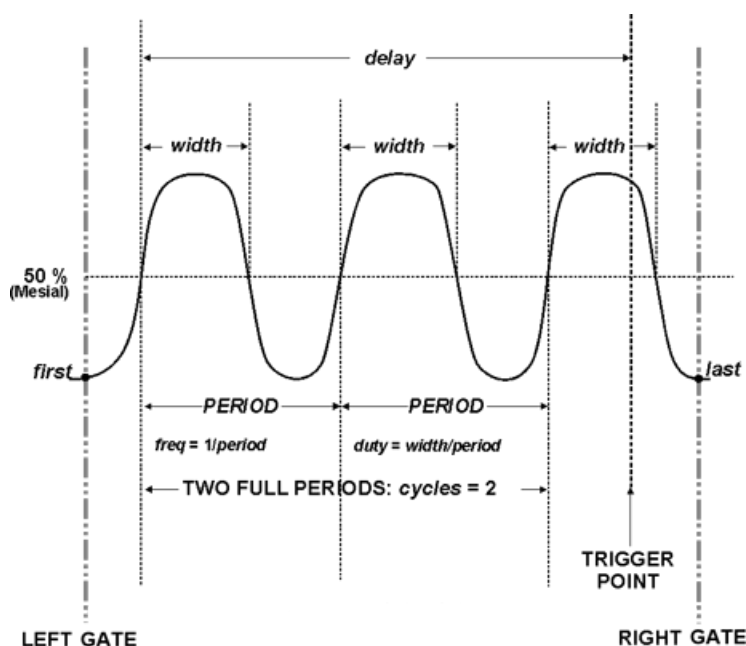
Falling Edge Duration

$$\frac{1}{M_f} \sum_{i=1}^{M_f} (Tf_i^{10} - Tf_i^{90})$$

Where M_r is the number of rising edges found, M_f the number of falling edges found, Tr_i^x the time when rising edge i crosses the $x\%$ level, and Tf_i^x the time when falling edge i crosses the $x\%$ level.

Determining Time Parameters

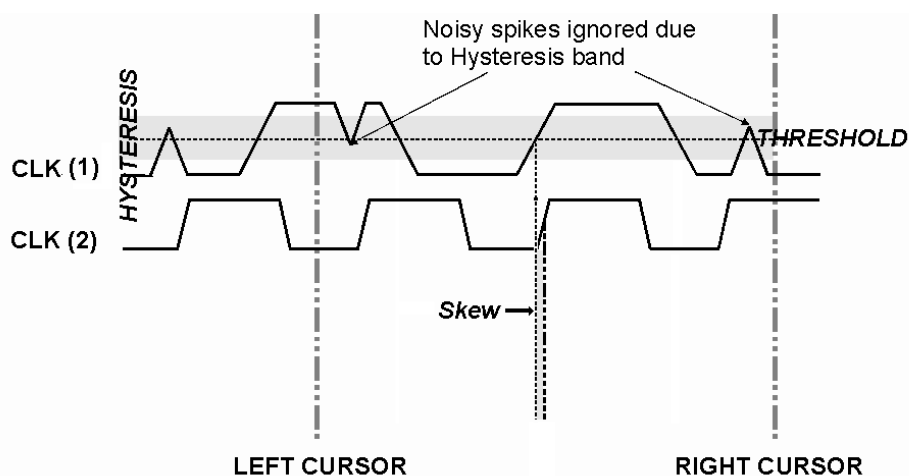
Time parameter measurements such as width, period and delay are carried out with respect to the mesial reference level, located halfway (50%) between the top and base reference lines, or with respect to the specified level for @level parameters.



Determining Differential Time Measurements

The instrument enables accurate differential time measurements between two traces: for example, propagation, setup and hold delays.

In the figure below, Skew measures the time interval separating the rising edge of Source1 from the rising edge of Source2.



Math

Math traces (F_n) display the result of applying a mathematical operation to a source trace. The output of a math function is always another trace, whereas the output of a measurement parameter is a tabular readout of the measurement.

Math can be applied to any channel (C_n), zoom (Z_n), or memory (M_n) trace. It can even be applied to another math trace, allowing you to chain operations (for example, trace F1 can show the average of C1, while trace F2 provides the integral of F1).

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 measure parameters and math functions will be available when using Measure and 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 function trace (F_n). Any additional settings required for the operator will appear on a subdialog at the right of the screen.

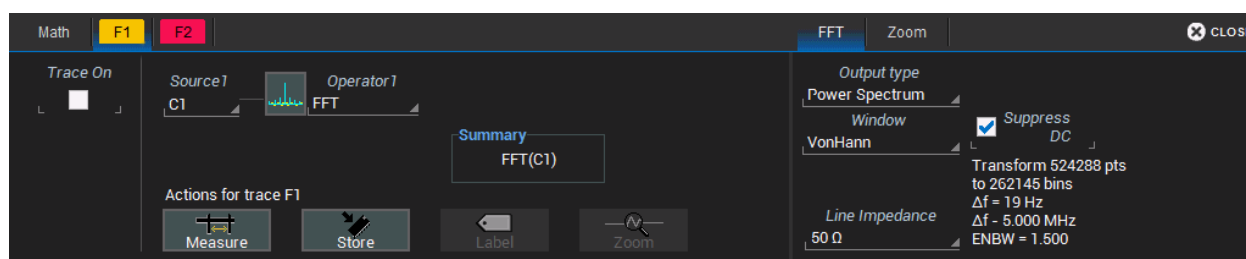
Single functions perform one operation on one or two input sources.

Dual functions chain two operations to arrive at a single result. This saves you the effort of having to chain two separate math functions. As with single functions, the number of sources required will vary based on the operation. You may need only one source for Operator1, but two for Operator2 (the result of the first operation counts as one source).

Setting Up New Functions

1. From the menu bar choose **Math > Math Setup**, or press the front panel **Math** button.
2. Choose a location by touching one of the **F_n** tabs.

If you know which function location you'll be using, you can select **F_n Setup** right from the Math menu.



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

- 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).
- 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.

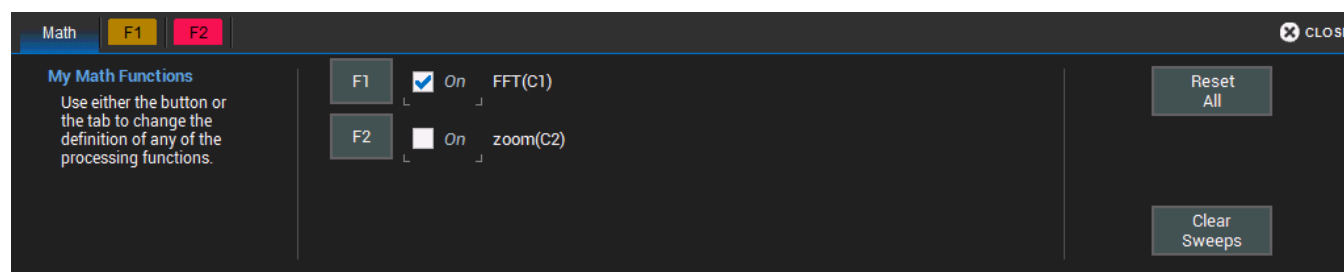
Adjusting Memory or Math Traces

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, touch the front panel **Math button**, or from the menu bar choose **Math > Math Setup**. Select the **On** checkbox next to each function you wish to display.

To modify a function, touch the **Fn button**.

To erase all functions from their locations, touch **Reset All**.

To restart the counter on cumulative functions (like Average), touch **Clear Sweeps**.

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. The valid range is 1 to 1,000,000 sweeps.



Tip: To quickly set up Continuous Averaging (only), access the channel setup dialog (Cn) and enter the number of sweeps to average in Averaging. The valid range is 1 to 1,000,000 sweeps.

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/AUTO to 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. Continuous averaging allows you to make adjustments to a system under test and to see the results immediately. The formula for both summed and continuous averaging is:

$$\text{new average} = (\text{new data} + \text{weight} * \text{old average}) / (\text{weight} + 1)$$

However, 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	$(\text{new data} + 4 * \text{old average}) / (4 + 1) = 1/5 \text{ new data} + 4/5 \text{ old average}$

In this way, for sweeps > 4 the importance of the old average begins to decrease exponentially.



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. However, it is more efficient concerning bandwidth and pass-band filtering.

Use ERes:

- On single-shot acquisitions, or where the data record is slowly repetitive (cases where you cannot use averaging).
- To reduce noise on noticeably noisy signals when you do not need to perform noise measurements.
- 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 the Instrument Enhances Resolution

The instrument's enhanced resolution 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. The enhanced resolution 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 increased by	-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

The choice of a spectral window is dictated by the signal's characteristics. 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.

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 (e.g., V · A becomes W).

Units of Measure

Following are the supported SI units of measure and the mnemonics used to represent them on the Rescale dialog. These same mnemonics can be used in remote control programs and customization scripts.



Note: Specify only the base unit in code, do not add prefixes. 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.



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/S ²
	inches/second ²	IN/S ²
	feet/second ²	FT/S ²
	standard gravity	GN
Pressure	Pascal	PAL
	bar	BAR
	atmosphere, technical	AT
	atmosphere, standard	ATM
	Torr	TORR
	pounds/square inch	PSI

Category	Unit	Mnemonic
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
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

Category	Unit	Mnemonic
Time	second	S
	minute	MIN
	hour	HOUR
	day	DAY
	week	WEEK
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

List of Standard Operators

The math operators included standard with your oscilloscope are listed below alphabetically.



Note: There may be additional operators available depending on the software options installed on the oscilloscope.

Operator	Definition
Absolute	For every point in the waveform the distance away from zero is calculated. For values greater than zero this is the same as the value. For values less than zero, the magnitude of this value 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: <i>(next sample value – current sample value) / (horizontal sample interval)</i>
Difference	For every point in the waveform, the value of Source2 is subtracted from the value of Source1. Source1 and Source2 must have the same horizontal units and scale and the same vertical units.
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: <i>(current sample value + next sample value) * (horizontal sample interval)</i> 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, the value of Source1 is divided by the value of Source2. Source1 and Source2 must have the same horizontal units and scale.
Reciprocal	For every point in the waveform the inverse is calculated using the formula: $1 / (\text{sample value})$
Rescale	For every point in the waveform the sample value is multiplied by the specified multiplier and then add to with the specified adder. 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 Sin(x)/x filter.
Square	For every point in the waveform, the square of the sample value is calculated.

Operator	Definition
Square Root	For every point in the waveform, the square root of the sample value is calculated.
Sum	For every point in the waveform, the value of Source1 is added to the value of Source 2. Source1 and Source2 must have the same horizontal units and scale and the same vertical units.
Track	Generates a waveform composed of parameter measurements that is time synchronous with the source waveform. The vertical units are those of the source parameter value and the horizontal units are seconds. Parameter values are posted at the sampling rate.
Trend	Produces a waveform composed of a series of parameter measurements 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 measurement.
Zoom	Produces a magnified trace of a selected portion of the input waveform. See Zooming Traces .

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

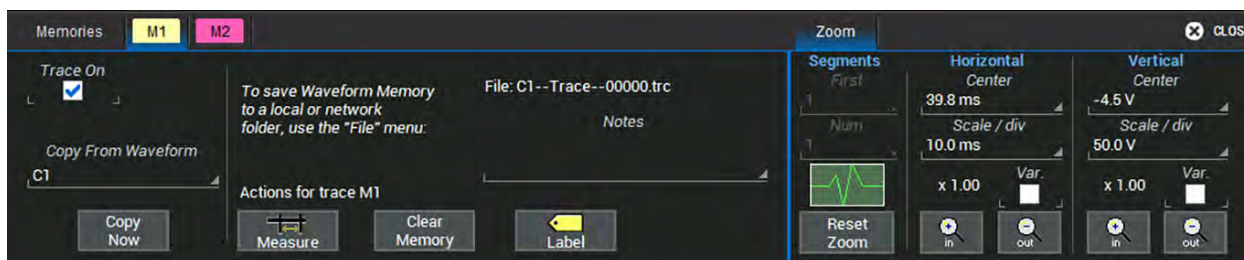


Tip: Try to choose an empty slot, as anything currently stored in that location will be overwritten. All memories will state if they are empty or an acquisition is stored there.

On oscilloscopes with OneTouch, touch the **Add New box** and choose **Memory** until you see an empty memory slot. Drag the descriptor box of the trace you wish to store onto the Mn descriptor box.

Or

1. Press the front panel **Mem button** or 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.

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: On WaveSurfer 3000/3000z oscilloscopes, 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.

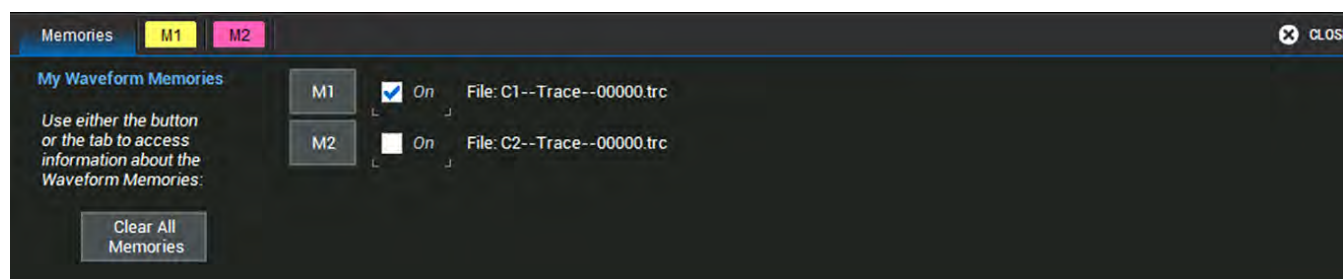
Access the Memories dialog by pressing the front panel **Mem** button or choosing **Math > Memory Setup**.

Check **On** next to the memory trace you wish to display. A description of the memory showing the source channel and creation time appears next to each **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.

Optional software packages may be purchased for specialized uses, such as power analysis. In most cases, these options are added to the Analysis menu. Documentation for software options can be found at [under Manuals > Software Options](#).

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.

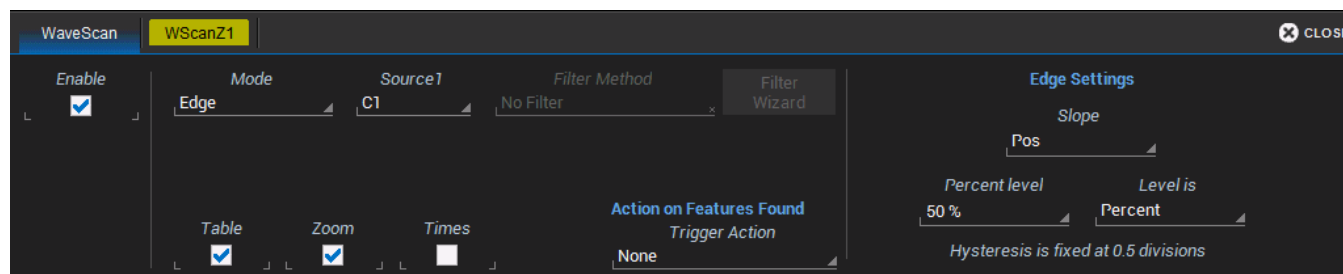
You customize the presentation by choosing different WaveScan displays, called [Scan Views](#). Optionally, set actions, such as stopping or beeping, to occur automatically when unusual events are found.



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**.
3. Check **Enable**.
4. Choose the **Source** waveform.
5. Choose the [Scan Mode](#) and enter values for any additional settings that appear at the right of the dialog based on your selection.
6. 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.
7. Optionally, choose an **Action** to trigger when an event that meets your scan criteria is found.
8. 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 X-Stream software, creating a new trace that is a magnified section of the source trace. A *Zn* 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.



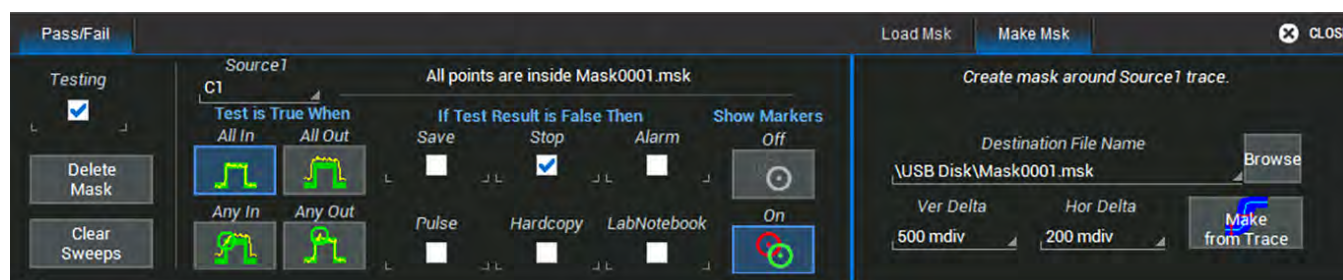
Note: Pass/Fail testing does not work with Sequence Mode acquisitions.

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. To use a saved **.msk** file, touch **File** and select the mask.

3. Check **View Mask** to display the mask over the trace.



Tip: Select the mask file and touch **Delete** to remove it from the directory.

Remove a Mask from the Display

Touch the **Delete Mask** button at the left of the Pass/Fail dialog.

Run Test

1. Make or load a mask.
2. 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.).
3. 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 **Hardcopy**(Print) setting
 - Create a **LabNotebook** Entry
4. Choose to turn **Markers Off** or **On**. When "On", points where the signal intersects the mask are marked by a red circle.
5. 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.

Save / Recall

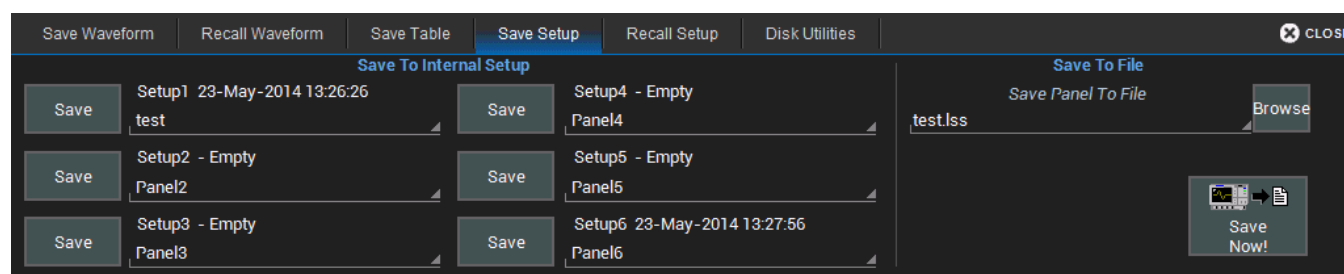
The Save/Recall dialog displays a series of shortcut buttons launching the various Save/Recall functions. You can use these buttons or the tabs to navigate to the other Save/Recall dialogs.

Save Setups

Save Setups allows you to quickly save up-to-six panel settings to internal storage, while Recall Setups restores them with a touch.

If desired, you can also save panel settings as an .LSS file, a different location, such as a USB drive. You can recall them from the same.

Choose **File > Save Setup...** from the menu bar.



Save Setup to Memory

1. Touch one of the **Setup** data entry controls and enter a name for the memory.
2. Touch the corresponding **Save** button directly to the left of the Setup field.

The save date/time is displayed above the **Setup** data entry control.

Save Setup to File

1. In **Save Panel to File**, touch **Browse** and navigate to the desired folder (Storage Card or USB Disk).



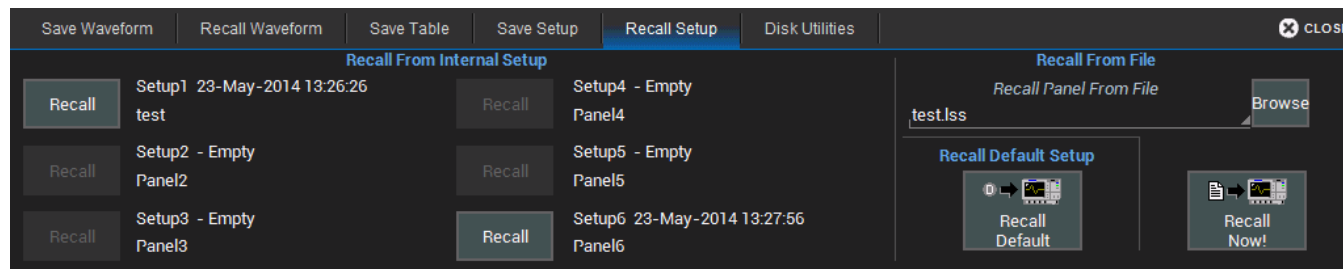
Tip: If the instrument is networked, you can touch Save Panel to File and enter the full Windows network address of another location in which to save the file. The instrument must have access to this directory. If this path remains on the Save Setups dialog when the oscilloscope is turned off, you will be asked for log on credentials to this directory when the oscilloscope is rebooted.

2. Enter a **File name**, or choose a existing file to overwrite. Touch **OK**.
3. On the Save Setups dialog, touch **Save Now!**

Recall Setups

Recall Setups restores setups saved to one of the internal memory locations, or enables you to import a setup file.

Choose **File > Recall Setup...** from the menu bar.



Recall Setup from Memory

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. In **Recall panel from file**, touch **Browse** and navigate to the desired folder.
2. Select the setup file and touch **OK**.
3. On the Recall Setups dialog, touch **Recall Now!**

Save Waveforms

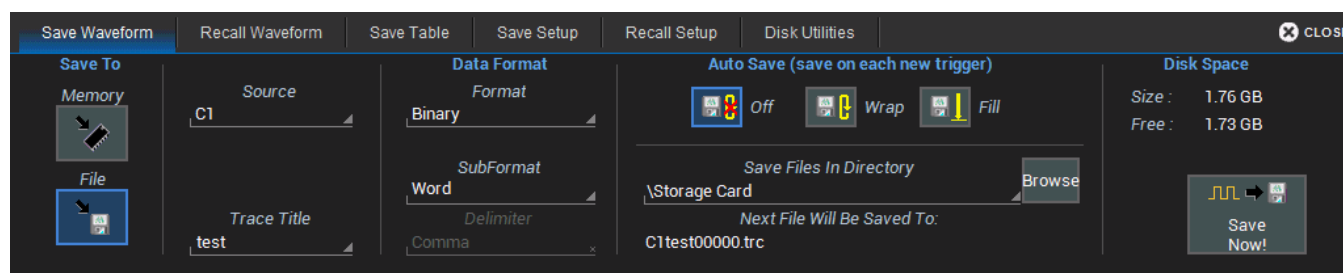
The Save Waveform function saves trace data to either an internal memory location, or to a trace file. The source waveform can be any type of trace; a channel, math function, zoom, or even another memory. Use Recall Waveform to restore these previously saved waveforms to the display.



Note: Only files saved in binary format (.TRC) can be recalled to the touch screen.

By default, trace files are saved to the MicroSD card, although you can choose another location, such as a USB drive. The file name is autogenerated from the <source trace><trace title><number in sequence> (e.g., C1test000001).

Choose **File > Save Waveform** from the menu bar.



Save Waveform To Memory

1. Touch **Memory**.
2. Choose the **Source** trace you are saving.
3. Choose the **Destination** location.
4. Touch **Save Now!**

Save Waveform To File

1. Touch **File**.
2. Choose the **Source** waveform.
3. Optionally, touch **Trace Title** to change the root file name of your waveforms.



Caution: Numbers placed at the end of this name are truncated because the instrument appends a sequence number to each file. Place numbers at the beginning, or place an alpha character after the number (e.g., XYZ32a).

4. Touch **Data Format** and select a file format:
 - **Binary**, Teledyne LeCroy's binary file format (.trc). Binary results in the smallest possible file size, and is necessary for recalling waveforms to Teledyne LeCroy instruments.



Note: Binary files can be converted to ASCII using Teledyne LeCroy utilities such as ScopeExplorer or WaveStudio.

- **ASCII** text file (.txt extension).
 - **MATLAB** text file (.dat extension).
 - **Excel** text file (.csv extension).
 - **MathCad** text file (.prn extension).
 - **Audio** .wav file.
5. Depending on your file format selection, you may also need to specify a **SubFormat**:
- **Word** (Binary) represents samples in the output file with 16 bits. Always use this options unless Byte mode is "pre."
 - **Byte** (Binary) represents samples in the output file with 8 bits. This option can result in a loss of output file 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.
6. If you selected **ASCII** format, also touch **Delimiter** and select a delimiter character from the pop-up menu.
7. In Save Files in Directory, touch **Browse** and navigate to the desired location (Storage Card or USB Disk). Touch **OK**.



Tip: If the instrument is networked, you can touch on Save Files in Directory and enter the full Windows network address of another location in which to save the file. The instrument must have access to this directory. If this path remains on the Save Waveform dialog when the oscilloscope is turned off, you will be asked for log on credentials to this directory when the oscilloscope is rebooted.

8. On the Save Waveform dialog, touch **Save Now!**

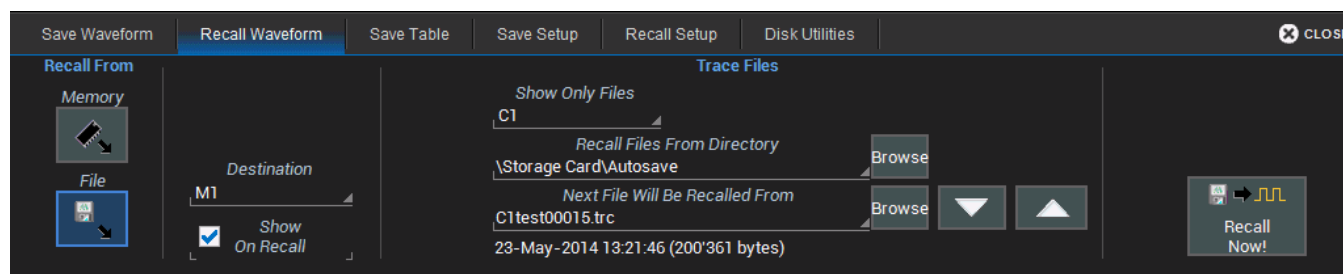
Recall Waveforms

Use Recall Waveform to restore previously saved waveform files to the display.



Note: Only files saved in binary format (.TRC) can be recalled to the touch screen.

Choose **File > Recall Waveform** from the menu bar.



Recall Waveform From Memory

1. Touch **Memory**.
2. Touch **Source** and choose a memory location from the **Select Source** pop-up.
3. Touch **Destination** and select a location into which to open the recalled memory.
4. Mark **Show on Recall** to display the trace on the grid.
5. Touch **Recall Now!**

Recall Waveform From File

1. Touch **File**.
2. Touch **Recall files from directory** and enter the path to the waveform folder, or touch **Browse** and navigate to the folder.
3. 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.



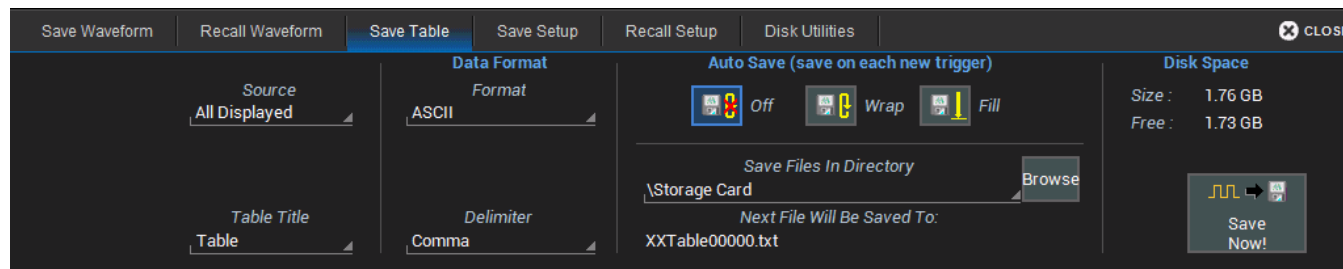
Note: The filter only applies to the files that will appear in the "Next File Will Be Recalled From..." list. It does not impact the Browse function.

4. Mark **Show on Recall** to display the trace on the grid.
5. Touch **Recall Now!**

Save Table Data

The Save Table function saves tabular measurement data displayed on screen to an Excel or ASCII file. By default, files are saved on the MicroSD card, although you can choose a USB drive.

Access the **Save Table** dialog by choosing **File > Save Table** from the menu bar.



1. Leave the default **Source** selection All Displayed.
2. Optionally touch **Table Title** and enter a new root file name.



Caution: Numbers you place at the end of this name are truncated because the instrument appends a sequence number to each file. Place numbers at the beginning, or place an alpha character after the number (e.g., XYZ32a).

3. Touch **Data Format** and choose from **ASCII** (.txt) or **Excel** (.csv) format.
4. If you selected **ASCII** format, also touch **Delimiter** and choose a character.
5. In Save Files in Directory, touch **Browse** and navigate to the desired folder (Storage Card or USB Disk). Select it and touch **OK**.



Tip: If the instrument is networked, you can touch Save Files in Directory and enter the full Windows network address of another location in which to save the file. The instrument must have access to this directory. If this path remains on the Save Table dialog when the oscilloscope is turned off, you will be asked for log on credentials to this directory when the oscilloscope is rebooted.

6. On the Save Table dialog, 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.

The Auto Save enables you to automatically store waveform and table data to a file that can be recalled to the oscilloscope later or saved permanently to external storage.

To automatically save data to a file after each new trigger, choose an **Auto Save** option on the Save Waveform and Save Table dialogs: **Wrap** (old files overwritten) or **Fill** (no files overwritten).



Caution: 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.

By default, waveform files and table data are stored on the MicroSD Card.

LabNotebook

The LabNotebook feature allows you to create and save Notebook Entries containing all setups, a capture of all displayed waveforms and waveform data, to which you may add custom annotations.

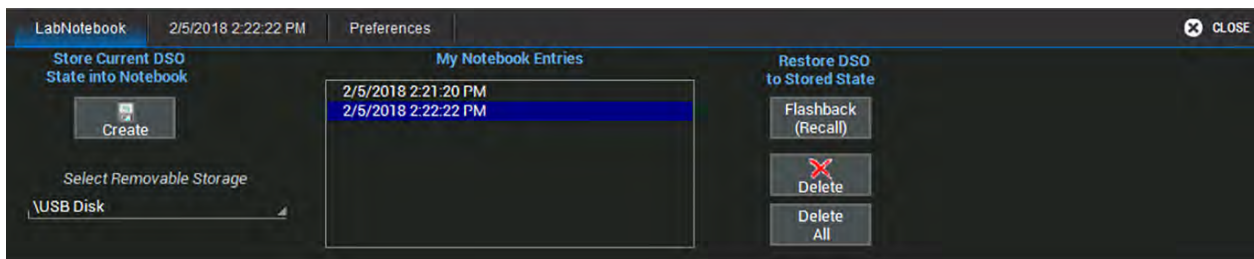
Notebook Entries are stored in an internal database and are available to be recalled to the touch screen at any time. A keyword filter makes it easy to find and recall a specific Notebook Entry.

The Flashback Recall feature instantly recalls the setups and waveforms stored with individual Notebook Entries, enabling you to restore the exact state of the instrument at a later date to perform additional analysis.

Create Notebook Entry

A Notebook Entry is a snapshot of the instrument at the moment it is taken: it captures the waveforms, their setups, and any measurements in process. As each new entry is created, it is added to the current database of Notebook entries. All entries are accessible from the LabNotebook dialog, from which they can be recalled to the screen through Flashback Recall.

1. Choose **File > LabNotebook** to open the LabNotebook dialog.



2. Touch **Create**.
3. Optionally, **Enter Report Title** and **Description**.

The default title is the date and time stamp.



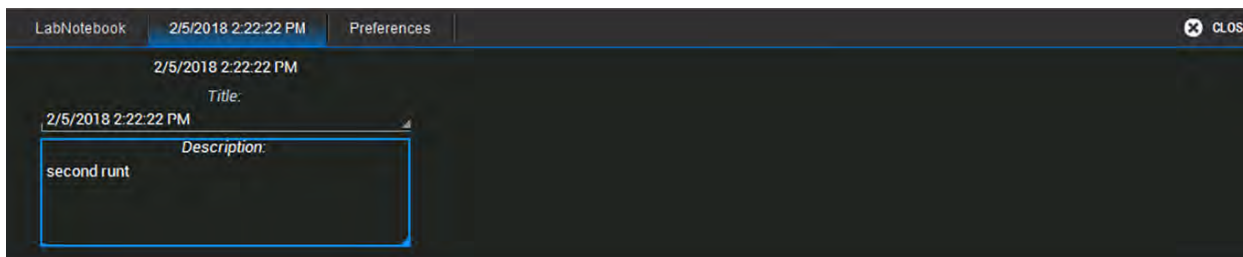
Note: By default, you will be prompted to title and annotate notebook entries as they are created. You can [configure LabNotebook preferences](#) so that these steps are skipped in order to streamline the creation process. To update entries at a later time, select the entry from the list of Notebook Entries, then open the tab of the same name that appears behind the LabNotebook dialog.

Manage Notebook Entries

First select the entry from the **My Notebook Entries** list.

Edit Notebook Entries

1. Select the entry from the **My Notebook Entries** list.
2. Go to the **second tab** labeled with the entry name.



3. Modify the **Title** or **Description**.

Delete Notebook Entries

Use the **Delete** button on the LabNotebook tab to remove selected Notebook Entries, or **Delete All** to clear all of **My Notebook Entries**.



Note: Unless you have previously backed up the notebook, deleted entries cannot be restored.

Flashback Recall

Once a Notebook Entry is made, you can recall it at any time using Flashback Recall. The recall includes waveforms and panel settings, so you can analyze the inputs that resulted in that capture.

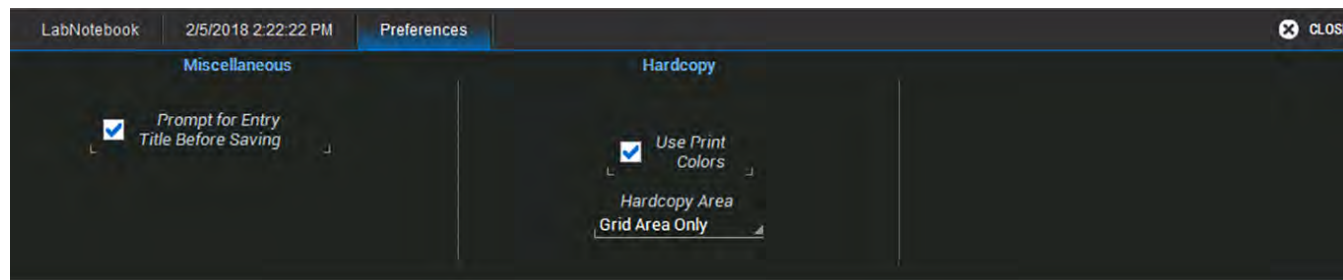
1. Choose **File > LabNotebook** to open the LabNotebook dialog.
2. Select the **Notebook Entry** from the list.
3. Touch the **Flashback Recall** button.
4. To exit Flashback Recall, touch the **Undo** button at the far right of the menu bar.

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

- **Persistence data** (although it is saved in with the Notebook Entry).
- **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 Recall 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.

LabNotebook Preferences

To modify the behavior of the LabNotebook tool, change settings on the LabNotebook **Preferences** dialog:



Prompt for Entry Title Before Saving will cause a pop-up for entering a custom Title and Description to appear when a new entry is created. You can elect to name notebook entries using only the date/timestamp by leaving this box unchecked.

Use Print Colors saves the waveforms on a white background. This option helps save ink/toner when printing entries.

Hardcopy Area determines how much of the screen image is included in the report: Grid Area Only, or entire DSO Window.

Utilities

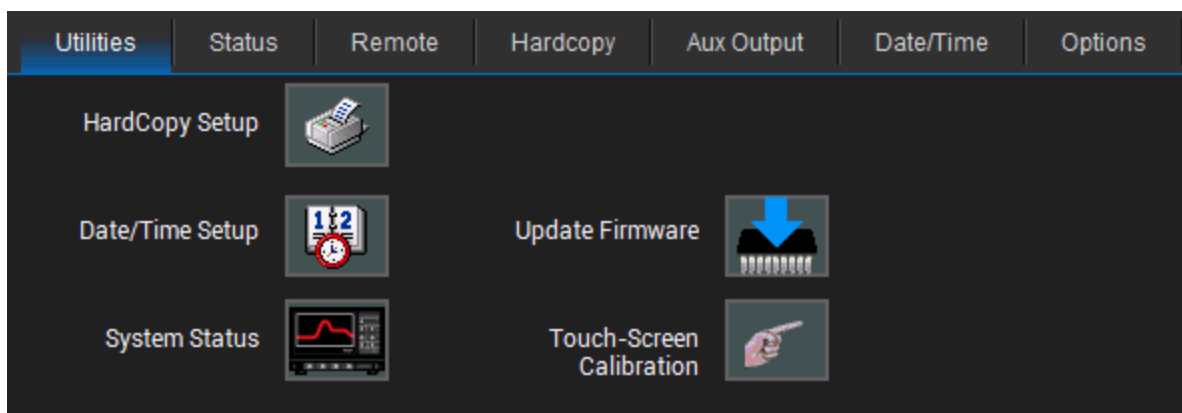
[Utilities](#) settings primarily control the instrument's interaction with other devices/systems.

[Preferences](#) settings, on the other hand, tend to control the appearance and performance of the oscilloscope application. Use these settings to personalize the behavior of the instrument.

[Disk Utilities](#) enable you to perform commonplace Windows file functions.

Utilities Dialogs

To access the Utilities dialogs, choose **Utilities > Utilities Setup...** from the menu bar.



[Hardcopy Setup](#), [Date/Time Setup](#) and [System Status](#) buttons open their corresponding dialogs.

There are also tabs linking to [Remote Control](#), [Aux Output](#), and [Options](#) settings.

Update Firmware opens a dialog for [installing the X-Stream application 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.

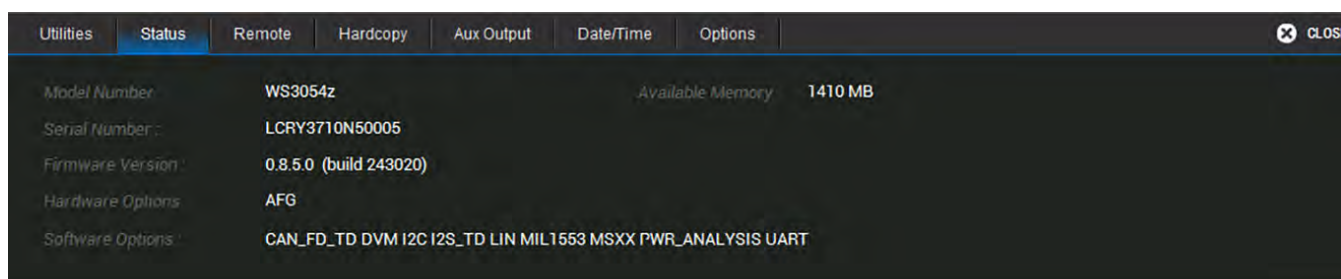
Status

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, also install Teledyne LeCroy's VICP drivers on the controller. These are included in the VICP Passport plug-in, available free from [www.teledynelecroy.com](#) under Oscilloscope Downloads > Software Utilities. The instrument uses Dynamic Host Configuration Protocol (DHCP) as its default addressing protocol, although you can manually assign a static IP address.
- **LXI (Ethernet).**
- **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: Full remote control setup requires installation and configuration of software on the controller, as well. See the [MAUI Oscilloscopes Remote Control and Automation Manual](#) for full instructions on making the remote connection and sending remote commands. The steps shown here are only those performed on the oscilloscope to enable the connection. A commented copy of the oscilloscope Waveform Template (.tpl) file is installed on the oscilloscope in C:\Program Files\LeCroy\XStream. You can open this ASCII file using any text editor, such as Windows Notepad, which is also installed on the oscilloscope. The template will help you better understand the MAUI architecture for transferring waveform data to and from the oscilloscope.

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.

- On the Ethernet Driver Settings dialog, choose **Specify an IP Address**.

If assigning a Name server, leave Obtain an IP address via DHCP.

- 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.

- 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.

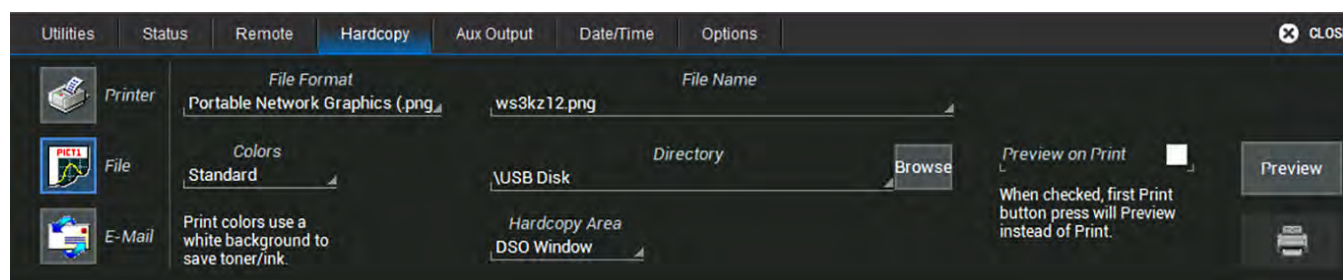


Note: These may be the credentials of any valid domain user (e.g., your own login); they do not have to be unique to the oscilloscope. Consult with your Network Administrator.

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

Hardcopy (Print)

Hardcopy settings control how the **Print** function behaves. Using either the Print button or the touch screen, Print captures an image of the touch screen display, but there are several options as to what it does next with the image, which are configured on the Utilities Hardcopy dialog. You can also set up a default print color scheme and capture area.



Send to Printer

Follow these steps to configure output to a printer.



Note: WaveSurfer 3000 oscilloscopes support PictBridge-compatible printers. Printers can be connected via LAN (Ethernet) or USB.

1. On the **Hardcopy** dialog, choose **Printer**.
2. Choose a page **Orientation**: portrait or landscape.
3. To print immediately, touch the **Printer icon** at the far right of the dialog.

Print to File

1. On the Hardcopy dialog, choose **File**.
2. Choose the output **File Format**.
3. Enter a **File Name**. This will form the basis of all image file names until changed.



Note: Numbers you add to the end of the file name will be truncated, as the instrument appends numbers to this name with each new file. To add your own identifying numbers, place them at the front of the name.

4. Optionally, enter the path to a new save **Directory** (the default folder is D:\Hardcopy), or touch the **Browse** button and navigate to the folder.

E-Mail

Follow this procedure to e-mail capture files to a preset address. The e-mail connection is set up in **Utilities > Preferences Setup > E-Mail**.

1. On the Hardcopy dialog, choose **E-Mail**.
2. Choose the attachment **File Format**.
3. If you wish to be able to include messages with the files as they are sent, check **Prompt for message to send with mail**.
4. To set up the e-mail connection, touch **Configure E-Mail Server and recipient**. This will take you to the [E-Mail Preferences](#) dialog.

Choose Color Scheme

To change the color of your Print output, touch the **Color** button and choose from:

- **Standard**(default) - prints objects on a black background, as they appear on the display.
- **Print** - prints objects on a white background. This option saves ink.

Set Print Area

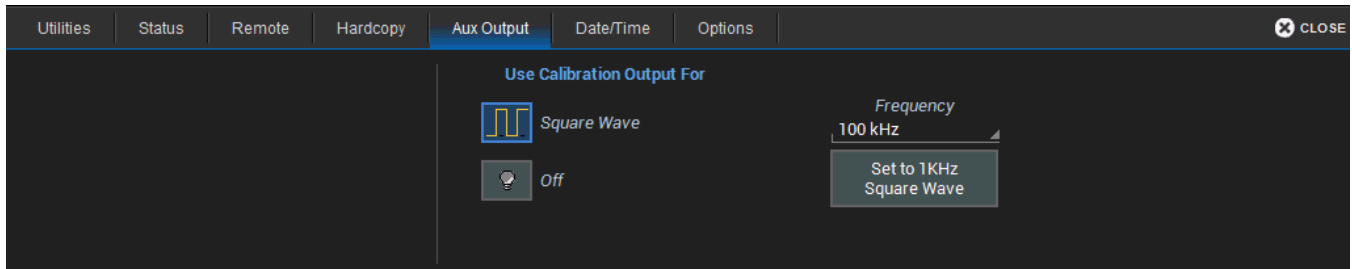
To limit which part of the touch screen is captured, touch **Hardcopy Area** and choose from:

- **Grid Area Only** - omits dialogs and menus and prints only the grids.
- **DSO Window** - prints the dialogs with the grids.

Aux Output

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

Use the Aux Output dialog to configure the calibration signal output from the **Cal Out** hook on the front of the oscilloscope. The calibration output voltage is 3V.



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.

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.

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.

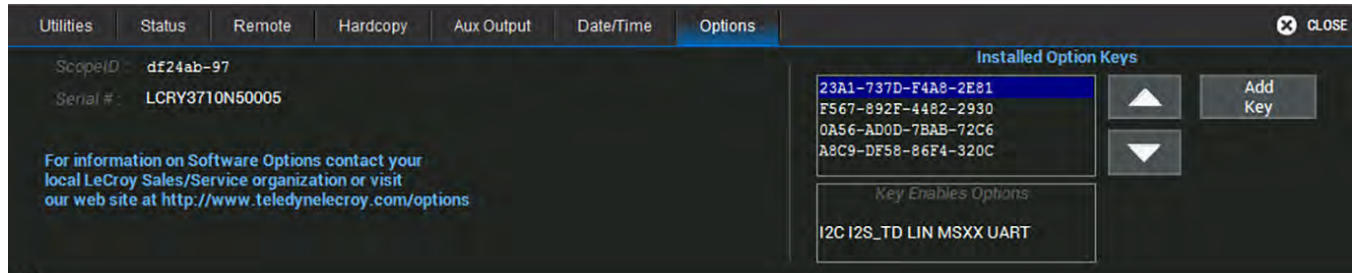
Time Zone

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.



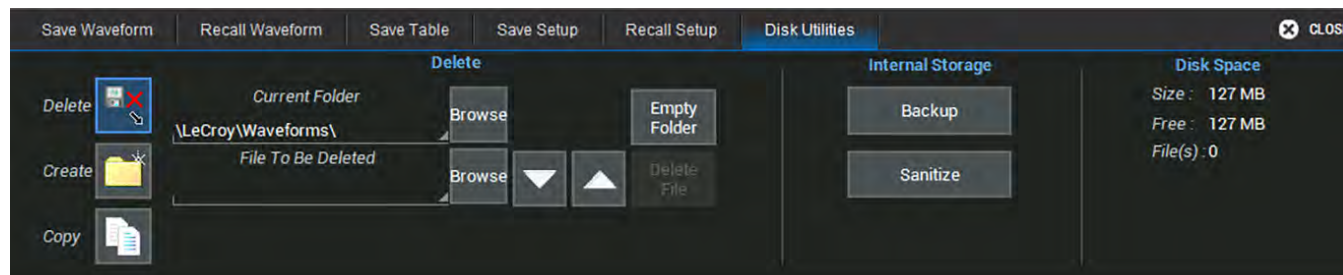
Tip: The Key Code is added to the list of Installed Option Keys. You can use the Up/Down buttons to scroll the list. The software option that each key activates is displayed below the list.

4. Choose **Yes** to restart the X-StreamDSO 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.



Delete a Single File

1. Touch the **Delete** button.
2. **Browse** to the current folder containing the file.
3. **Browse** to the file to be deleted, or use the **Up** and **Down** arrow buttons to scroll through the files in the folder.
4. With the desired file selected, touch **Delete File**.

Delete All Files in a Folder

1. Touch the **Delete** button.
2. **Browse** to the current folder containing the file.
3. With the desired folder selected, touch **Empty Folder**.

Create a New Folder

1. Touch **Create**.
2. Touch **Current folder** and provide the full path to the new folder, including the folder name.
3. Touch **Create Folder**.

Copy Folder/File

1. Touch **Copy**.
2. **Browse** to and select the folder or **File To Be Copied**.
3. **Browse** to and select the **Destination Folder** for the copy.
4. Touch **Copy Folder** or **Copy File**.

Back Up Files

Touch **Backup** to back up the entire contents of the hard drive 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 (MicroSD Card). Otherwise, Storage Card is the default.

Sanitize Instrument

Touch **Sanitize** to delete all user data (setups, waveform files, screen captures, LabNotebook entries, etc.) from the oscilloscope hard drive.

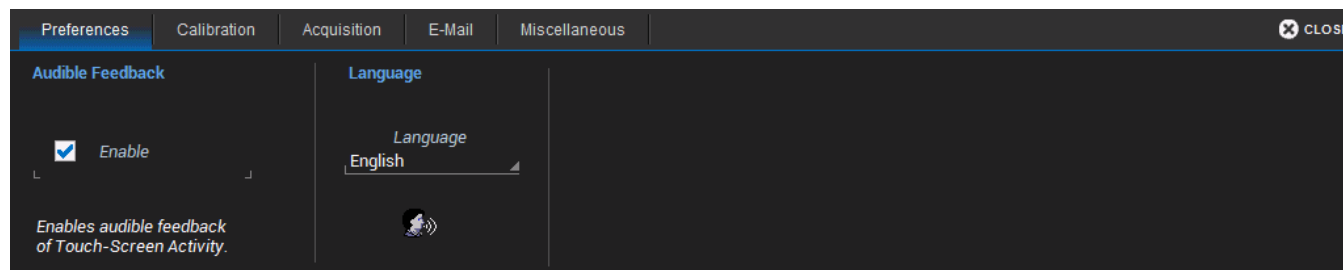


Note: The Sanitize function does not affect the MicroSD Card. The card can be removed before or after sanitizing for storage in a secure facility.

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 also called "non-volatile," because they are not lost and do not change when the oscilloscope is restarted or a setup panel is recalled.

Access the Preferences dialogs by choosing **Utilities > Preference Setup...** from the menu bar.



Power on AC will turn on the oscilloscope and launch the software as soon as you connect to AC power, without having to first press the Power button.

Audible Feedback controls the instrument's audio output. Select this box to hear a beep each time you touch a screen or front panel control.

Language sets the language used on the touch screen display.

Font Size changes the size of the text that appears on the touch screen display.

Performance settings let you optimize performance for either **Analysis** (speed of acquisition and calculation) or **Display** (speed of update/refresh). For example, if you are concerned with persistence or averaging, you might optimize for Analysis, giving higher priority to waveform acquisition at the expense of display update rate. Choices are presented as a spectrum.

Calibration

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:

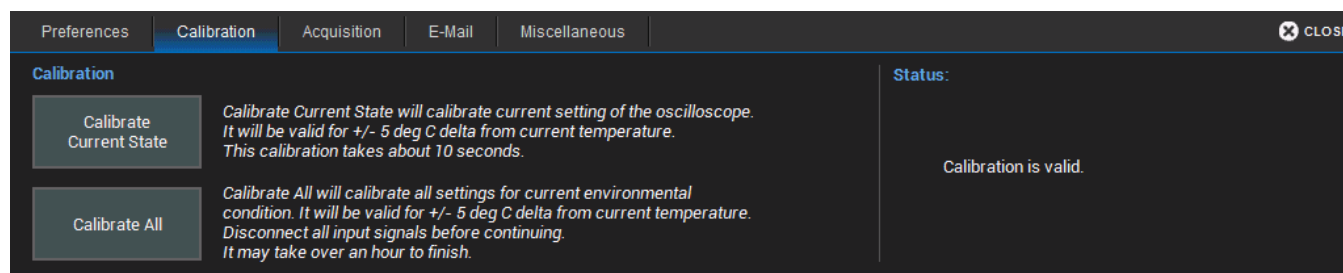
- It is used in temperatures that differ from the previous calibration temperature by more than 5° C.

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

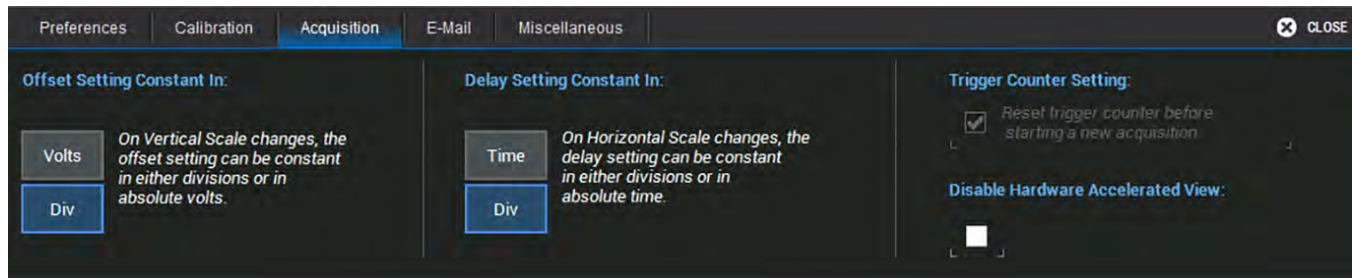
Calibrate Current Setting—the oscilloscope is calibrated at the current vertical and horizontal setting. This calibration is valid for this setting for the current temperature $\pm 5^{\circ}\text{C}$ and takes under 30 seconds.



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. To access them, choose **Utilities > Preference Setup** and 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: The Offset value is always in the vertical unit, and the Delay is always in time. However, whenever Div is selected, these 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.

E-Mail Preferences

Use the E-mail dialog settings to configure e-mail on the instrument.

Preferences Calibration Acquisition **E-Mail** Miscellaneous CLOSE

E-Mail Server Configuration

SMTP Server
domino.lecroy.com

Originator Address (From:)
scope.name@company.com

Default Recipient Address (To:)
your.colleague@company.com

Overview of E-Mail mode

SMTP uses a Simple Mail Transfer Protocol server directly. In many cases no account is required.

Send Test Mail

Under **E-Mail Server**, select the protocol used by your network:

- **MAPI** (Messaging Application Programming Interface) is the Microsoft interface specification that allows different messaging and workgroup applications (including e-mail, voice mail, and fax) to work through a single client, such as the Exchange client. MAPI uses the default Windows e-mail application.
- **SMTP** (Simple Mail Transfer Protocol) is a TCP/IP protocol for sending messages from one computer to another through a network. This protocol is used on the Internet to route e-mail. In many cases no account is needed.

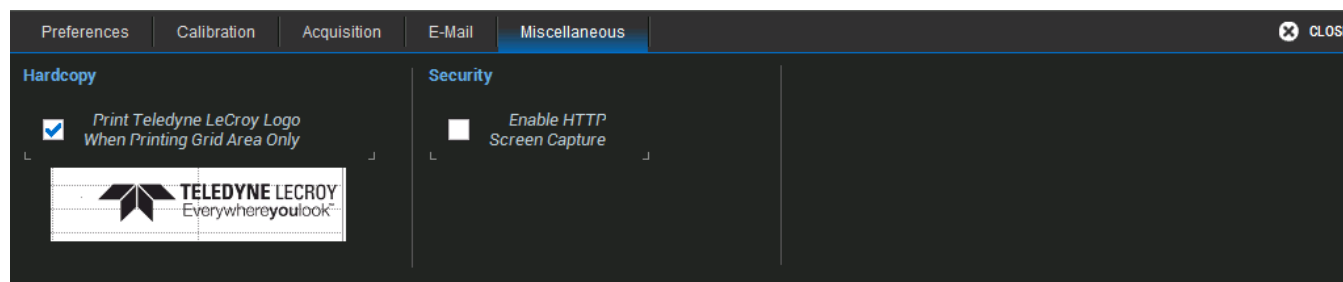
If you chose **SMTP**, touch **SMTP Server** and enter the network address of your mail server:

- Touch **Originator Address (From:)** and enter the instrument's e-mail address.
- Touch **Default Recipient Address (To:)** and enter the recipient's e-mail address.

Use **Send Test Mail** to send a confirmation message to ensure proper e-mail configuration.

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.

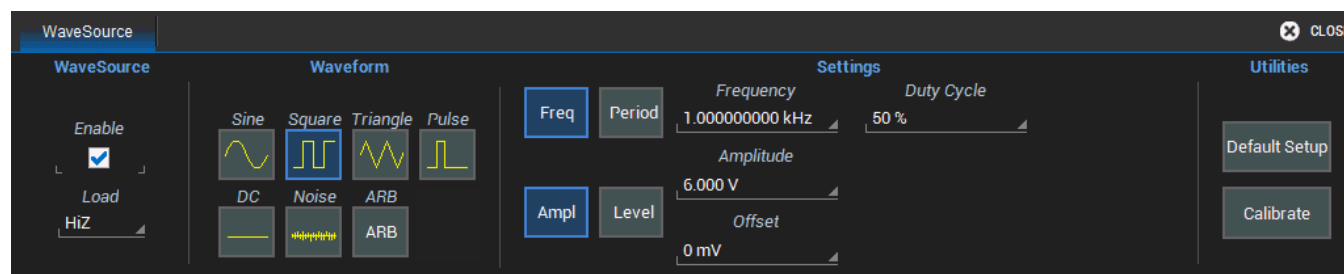
Check **Enable HTTP Screen Capture** to enable remote capture of the touch screen display over a network. This setting is required to use the instrument with the WaveStudio software.

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 WaveSurfer 3000.

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.

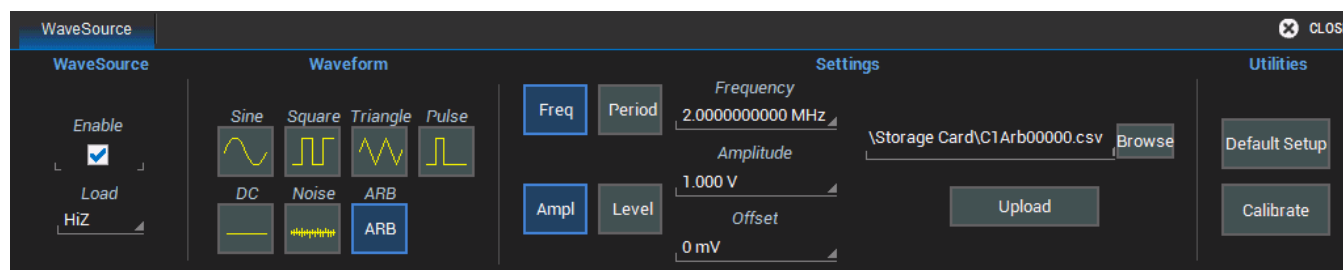
6. Check **Enable** to begin output.

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 (WS3K-DVM) 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.

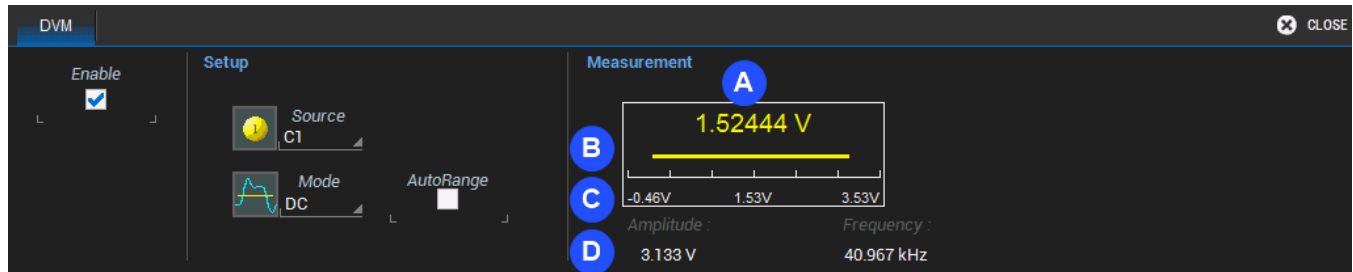
DVM Readout

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
 - DC RMS
 - AC RMS
 - 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.

Fuse Replacement

Disconnect the power cord before inspecting or replacing the fuse. Open the fuse holder (located at the rear of the instrument below the AC power inlet) using a small, flat-bladed screwdriver. Replace the old fuse with a new 5 x 20 mm T-rated 3.15 A/250 V fuse. Close the fuse holder before powering on.



Caution: For continued fire protection at all line voltages, replace the fuse with one of the specified type and rating only. Always disconnect the power cord before replacing the fuse.

Touch Screen Calibration

Periodically calibrate the touch screen to maintain its accuracy and responsiveness. We recommend that you use a stylus rather than your finger for this procedure.

1. From the menu bar, choose **Utilities > Utilities Setup**.
2. On the Utilities main dialog, touch **Touch-Screen Calibration**.
3. Following the prompts, touch as close as possible to the center of each cross that appears on the screen until the calibration sequence is complete.

Restart/Reboot Instrument

To restart the oscilloscope application, choose **File > Exit** from the menu bar, then touch the **Start DSO** desktop shortcut.



Note: You will generally need to exit and restart the application after adding new options keys.

To reboot the instrument, which includes restarting the OS:

1. From within the oscilloscope application, choose **File > Shutdown**.
2. Wait 10 seconds after the oscilloscope has fully shut down, then press the **Power button** on the front of the instrument.

WaveSurfer 3000 Firmware Update

Teledyne LeCroy frequently releases free firmware updates for X-Stream model oscilloscopes containing new product features and bug fixes. The X-Stream 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 MicroSD card.

1. Visit our download page at [www.lecroy.com/downloads](#) and click the link to **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.**

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, EN 61326-2-1:2013 EMC requirements for electrical equipment for measurement, control, and laboratory use. ¹

Electromagnetic Emissions:

EN 55011:2010, Radiated and Conducted Emissions Group 1, Class A ^{2 3}

EN 61000-3-2/A2:2009 Harmonic Current Emissions, Class A

EN 61000-3-3:2008 Voltage Fluctuations and Flickers, Pst = 1

Electromagnetic Immunity:

EN 61000-4-2:2009 Electrostatic Discharge, 4 kV contact, 8 kV air, 4 kV vertical/horizontal coupling planes ⁴

EN 61000-4-3/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/A1:2010 Electrical Fast Transient/Burst, 1 kV on power supply lines, 0.5 kV on I/O signal data and control lines ⁴

EN 61000-4-5:2006 Power Line Surge, 1 kV AC Mains, L-N, L-PE, N-PE ⁴

EN 61000-4-6:2009 RF Conducted Electromagnetic Field, 3 Vrms, 0.15 MHz - 80 MHz

EN 61000-4-11:2004 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 11:2011 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 Power Supply Circuits: 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.
- Unit: Pollution Degree 2, 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 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 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

Restriction of Hazardous Substances (RoHS)

This instrument and its accessories conform to the 2011/65/EU RoHS2 Directive.

ISO Certification

Manufactured under an ISO 9000 Registered Quality Management System.

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.

Intellectual Property

All patents pertaining to the WaveSurfer 3000/3000z can be found on our website at: