TELEDYNE LECROY
Everywhereyoulook™

Operator's Manual

WaveSurfer 10 Oscilloscopes

WaveSurfer 10 Oscilloscopes Operator's Manual

924609 Rev A
October 2014

Contents

Welcome

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

The manual is arranged in the following manner:

- **Safety** contains important precautions and information relating to power and cooling.
- The sections from **Start Up** through **Maintenance** cover everything you need to know about the operation and care of the oscilloscope.

Documentation for software options is available from the Teledyne LeCroy website at teledynelecroy.com. Our website maintains the most current product specifications and should be checked for frequent updates.

Remember...

When your product is delivered, verify that all items on the packing list or invoice copy have been shipped to you. Contact your nearest Teledyne LeCroy customer service center or national distributor if anything is missing or damaged. We can only be responsible for replacement if you contact us immediately.

Thank You

We truly hope you enjoy using Teledyne LeCroy's fine products.

Sincerely,

David C. Graef

Teledyne LeCroy
Vice President and Chief Technology Officer

Safety Instructions

Observe these instructions to keep the instrument operating in a correct and safe condition. You are required to follow generally accepted safety procedures in addition to the precautions specified in this section. **The overall safety of any system incorporating this instrument is the responsibility of the assembler of the system.**

Symbols

These symbols appear on the instrument's front and rear panels or in its documentation to alert you to important safety considerations:

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.

High voltage. Risk of electric shock or burn.

Ground connection.

Alternating current.

Standby power (front of instrument).

Precautions

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

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 or test leads while they are connected to a voltage source.

Observe all terminal ratings. Do not apply a voltage to any input (C1-C4 or EXT) that exceeds the maximum rating of that input. Refer to the front of the oscilloscope for maximum input ratings.

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

Use indoors only.

Keep product surfaces clean and dry. See Cleaning in the Maintenance section.

Do not block the cooling vents. Leave a minimum six-inch (15 cm) gap between the instrument and the nearest object. Keep the underside clear of papers and other objects.

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

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 impairment due to hazardous live electrical quantities. Cease operation immediately and sequester the instrument from inadvertent use.

Operating Environment

Temperature: 5 to 40° C.

Humidity: Maximum relative humidity 80 % for temperatures up to 31° C, decreasing linearly to 50% relative humidity at 40° C.

Altitude: Up to 3,000 m at or below 25° C.

Cooling

The instrument relies on forced air cooling with internal fans and vents. Take care to avoid restricting the airflow to any part. Around the sides and rear, leave a minimum of 15 cm (6 inches) between the instrument and the nearest object. The feet provide adequate bottom clearance.

CAUTION. Do not block cooling vents. Always keep the area beneath the instrument clear of paper and other items.

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.

Power

AC Power

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

Power Consumption

Maximum power consumption with all accessories installed (e.g., active probes, USB peripherals) is 340 W (340 VA). Power consumption in standby mode is 10 W.

Ground

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

WARNING. 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.

Start Up

Carrying and Placing the Oscilloscope

The oscilloscope's case contains a built-in carrying handle. Lift the handle away from the oscilloscope body, grasp firmly and lift the instrument. Always unplug the instrument from the power source before moving it.

Place the instrument where it will have a minimum 15 cm (6 inch) clearance from the nearest object. Be sure there are no papers or other debris beneath the oscilloscope or blocking the cooling vents.

CAUTION. Do not place the instrument so that it is difficult to reach the power cord in case you need to quickly disconnect from power.

Positioning the Feet

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

Powering On/Off

Press the **Power button** at the lower, left front of the oscilloscope to switch on the instrument; press it again to switch into Standby mode (reduced power). The oscilloscope application software loads automatically when you use the Power button.

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

CAUTION. Do not power on or calibrate the oscilloscope with a signal attached.

Always use the Power button or the **File > Shutdown** menu option to execute a proper shut down process and preserve settings before powering down. Do not power off by pulling the power cord from the socket or shutting off a connected power strip without first shutting down properly.

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

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

Software Activation

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

Firmware

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

teledynelecroy.com/support/softwaredownload.

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 on the oscilloscope. See Adding an Option Key for instructions on activating optional software packages.

Front Input/Output Panel

A. **Power button**.

B. **Channel inputs 1-4** for analog signals.

C. **EXT** to input an external trigger.

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.

Analog Inputs

A series of BNC connectors arranged on the front of the are used to input analog signal on Channels 1-4.

Channel connectors use the ProBus interface. The ProBus interface contains 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 MΩ input impedance and control for a wide range of probes.

The interfaces power probes and completely integrate the probe with the oscilloscope channel. Upon connection, the probe type is recognized and some setup information, such as input coupling and attenuation, is performed automatically. This information is displayed on the Probe Dialog, behind the Channel (C*x*) dialog. System (probe plus oscilloscope) gain settings are automatically calculated and displayed based on the probe attenuation.

Probes

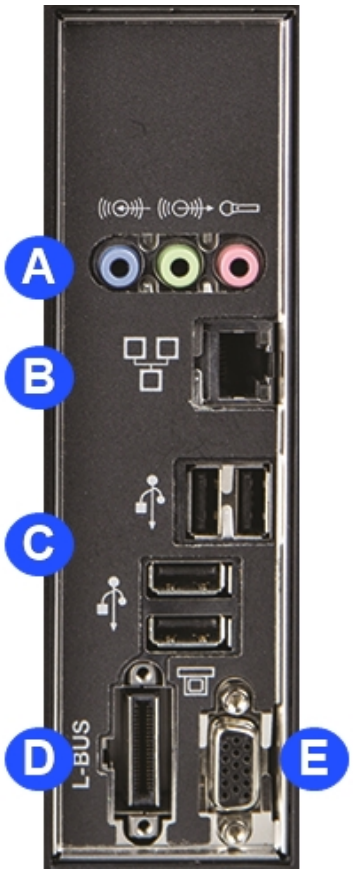
WaveSurfer 10 oscilloscopes are compatible with the included passive probes and all Teledyne LeCroy ProBus active probes that are rated for the oscilloscope's bandwidth. Probe specifications and documentation are available at teledynelecroy.com/probes.

The passive probes supplied with your oscilloscope 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.

Digital Inputs

The WaveSurfer 10 is compatible with the MS-250 and MS-500 Mixed Signal hardware options for input of up-to-36 lines of digital data. For instructions, see the product documentation available from teledynelecroy.com.

Side Input/Output Panel



A. Audio Input/Output **Line-In**, **Speaker**, and **Mic** jacks connect the oscilloscope to external audio devices.

B. **Ethernet Port** connects the oscilloscope to networks.

C. **USB Ports** (4) allow you to connect external USB devices, such as storage drives.

D. **L-BUS** connector interfaces the oscilloscope with the MS-250 or MS-500 external MSO module.

E. **VGA**connector sends video output to external monitors.

Connecting to Other Devices/Systems

Make all desired cable connections. After start up, configure the connections using the menu options listed below. More detailed instructions are provided later in this manual.

Power

Connect the line cord rated for your country to the AC power inlet on the back of the instrument, then plug it into a grounded AC power outlet. (See Power and Ground Connections in General Safety Information.)

LAN

The instrument accepts DHCP network addressing. Connect a cable from the Ethernet port on the side panel to a network access device.

To assign the oscilloscope a static IP address, go to Utilities > Utilities Setup > Remote and choose Net Connections from the Remote dialog. Use the standard Windows networking dialogs to configure the device address.

Go to Utilities > Preference Setup > Email to configure email settings.

USB Peripherals

Connect the device to a USB port on the front or side of the instrument.

Printer

The oscilloscope supports USB printers compatible with the oscilloscope's Windows OS. Connect the printer to any host USB port. Go to Utilities > Utilities Setup > Hardcopy to configure printer settings.

External Monitor

Connect the monitor cable to the VGA output on the sideof the instrument. Minimize the oscilloscope application and use the Windows controls to configure the display. Configure the oscilloscope as the primary monitor and be sure to extend, not duplicate, the display.

External Controller

Connect a USB-A/B cable from the instrument to the controller, or connect both to the same network using an Ethernet connection. Go to Utilities > Preference Setup > Remote to configure remote control.

Other Auxiliary Device

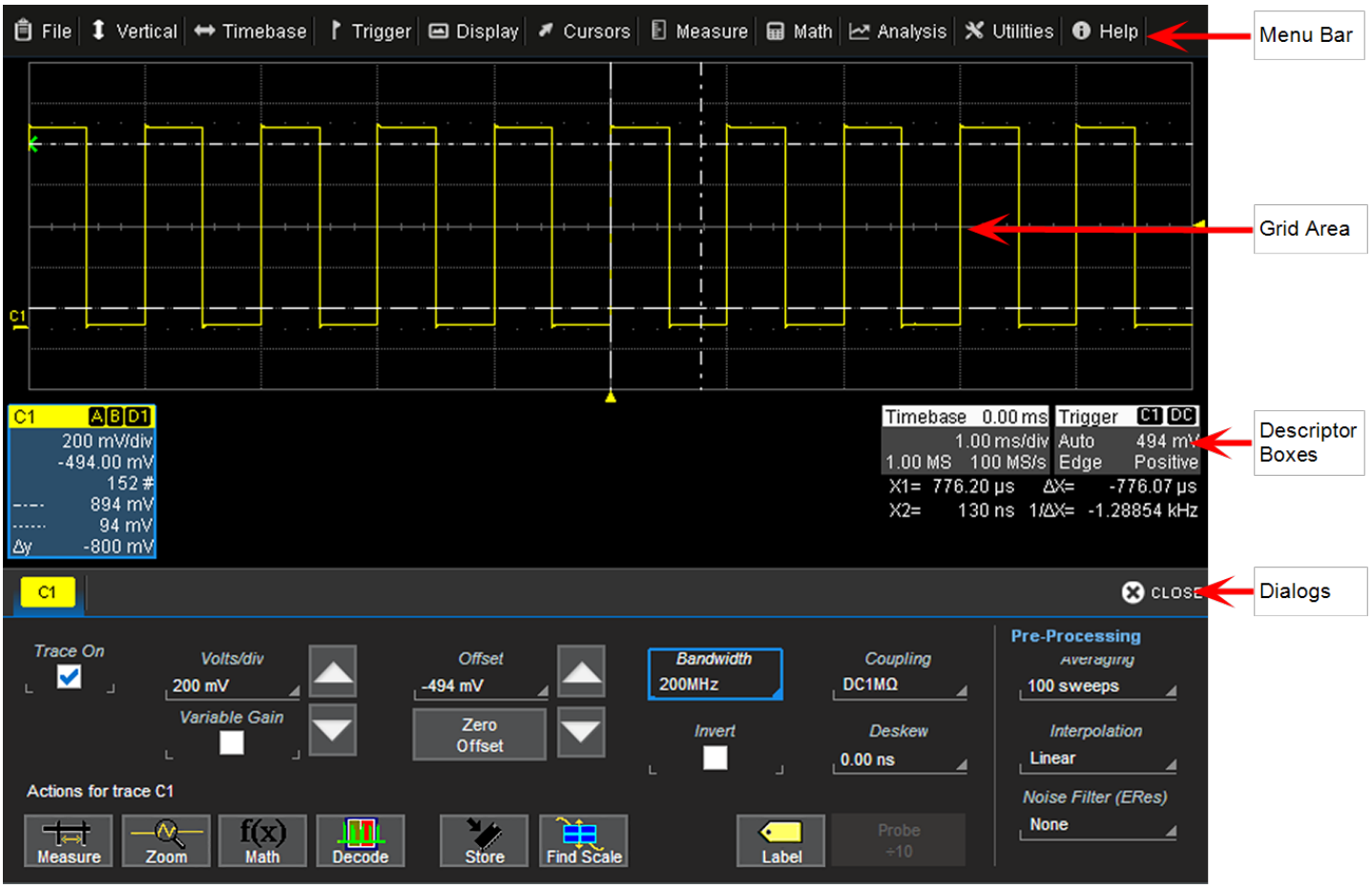
Connect a BNC cable from Aux Out on the back of the instrument to the other device. Go to Utilities > Utilities Setup > Aux Output to configure the output.

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 the stylus to touch, double-touch, touch-and-drag, or draw a selection box. Many controls that display information also work as “buttons” to access other functions.

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

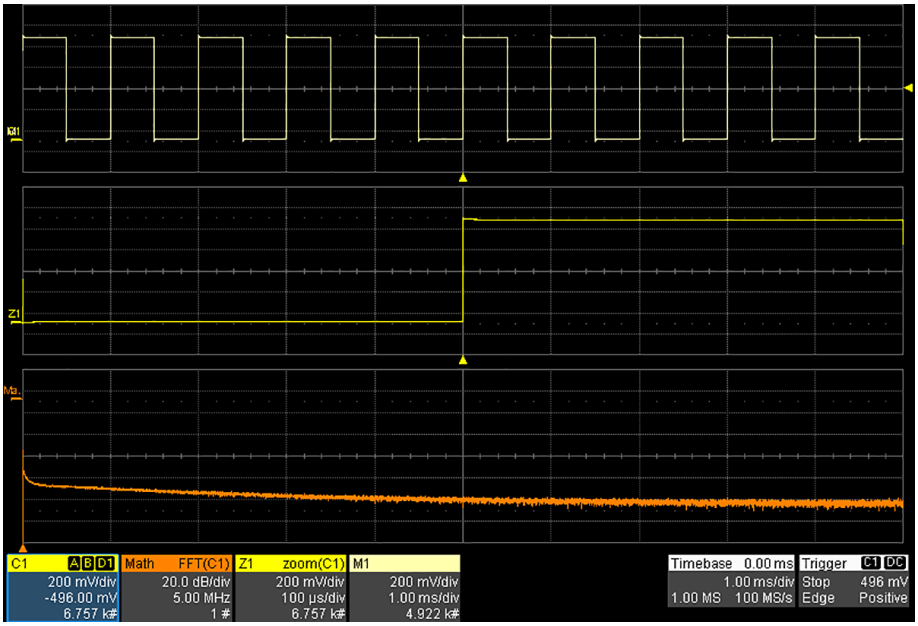
The top of the window contains a complete menu of oscilloscope functions. Making a selection here changes the dialogs displayed at the bottom of the screen.

Many common oscilloscope operations can also be performed from the front panel or launched via the Descriptor Boxes. However, the menu bar is the best way to access dialogs for Save/Recall (File) functions, Display functions, Status, LabNotebook, Pass/Fail setup, and Utilities/Preferences setup.

Grid Area

The grid area displays the waveform traces. Every grid is 8 Vertical divisions and 10 Horizontal divisions. The value of Vertical and Horizontal divisions depends on the Vertical and Horizontal scale of the traces that appear on the grid.

By default (Auto Grid mode), the grid area will automatically divide up to three times to display channel, zoom and math traces on different grids. Regardless of the number of grids, every grid always shows the same number of Vertical levels. Therefore, absolute Vertical measurement precision is maintained.



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

Adjusting Grid Brightness

You can adjust the brightness of the grid lines. Go to **Display > Display Setup** and enter 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.

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

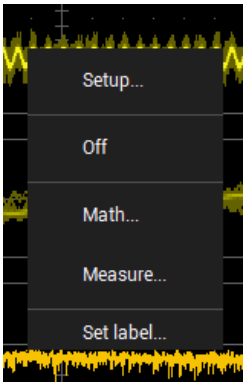
Pre/Post-trigger Delay, a small arrow to the bottom left or right of the grid, indicates that a pre- or post-trigger Delay has shifted the Trigger Position indicator to a point in time not displayed on the grid. All trigger Delay values are shown on the Timebase Descriptor Box.

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.

Various **Cursor lines** appear over the grid to indicate specific voltage and time values on the waveform. Touch-and-drag cursor indicators to quickly reposition them.

Grid Context Menu



Quickly touch a trace, or touch-and-hold the trace descriptor box, to open a pop-up menu with various actions such as turning on/off the trace, placing a label, or applying math and measurements.

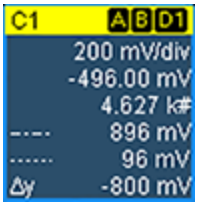
Descriptor Boxes

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; the box will be highlighted. Touch it a second time to open the trace setup dialog.

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



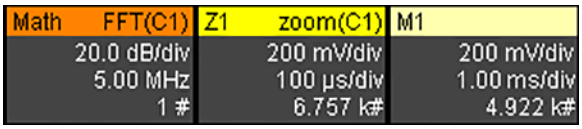
Channel trace descriptor boxes correspond to analog signal inputs. They show (clockwise from top left): Channel Number, Pre-Processing List, Coupling, Gain Setting, Offset Setting, Sweeps Count (when Averaging), and Vertical Cursor positions. Codes are used to indicate pre-processing that has been applied to the input. The short form is used when several processes are in effect.

Preprocessing Symbols on Descriptor Boxes

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

Other Trace Descriptor Boxes

Similar descriptor boxes appear for math, zoom (Zx), and memory (Mx) traces. These descriptor boxes show any Horizontal scaling that differs from the signal Timebase. Units will be automatically adjusted for the type of trace.

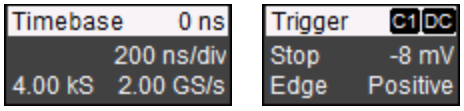


NOTE: On WaveSurfer 10 oscilloscopes with the WS10-ADT option installed, there will be two math functions, labeled F1 and F2 on the descriptor boxes and on the Math setup dialogs.

Timebase and Trigger Descriptor Box

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

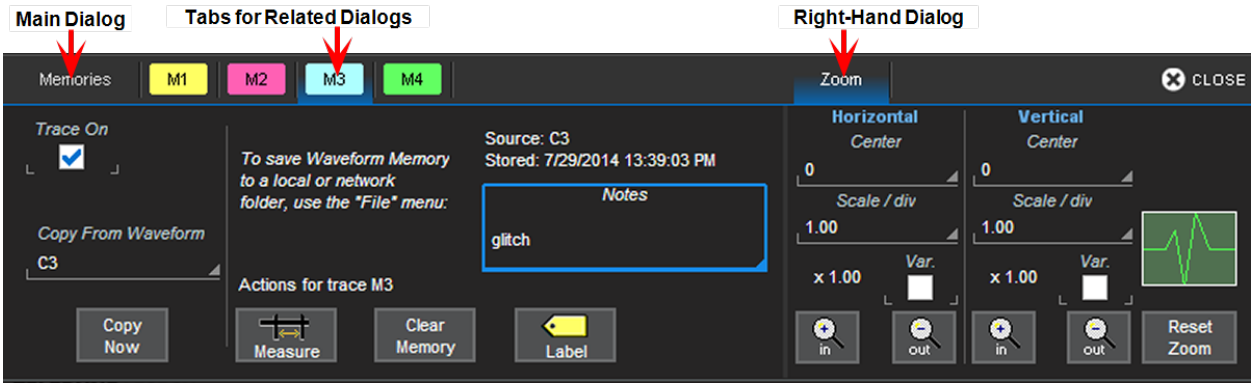
Trigger descriptor box shows: (clockwise from top right) Trigger Source and Coupling, Trigger Level (V), Slope, Trigger Type, Trigger Mode.



Setup information for Horizontal cursors, 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 setup option. For convenience, related dialogs appear as a series of tabs behind the main dialog. Touch the tab to open the dialog.



Right-Hand Dialogs

At times, your selections will require more settings than normally appear (or can fit) on a dialog, or the task commonly invites further action, such as zooming a new trace. In that case, sub-dialogs will appear to the right-side of the main dialog. These right-hand dialog 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 apply common actions without having to leave the underlying set up dialog. They 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 serial data decoders can be configured and applied. This button is only active if you have decoder 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 automatically performs a vertical scaling that fits the waveform into the grid.

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

Turning On/Off Traces

Analog Traces

From the menu bar, choose **Vertical > Channel <#> Setup** to turn on the trace. To turn it off, clear the **Trace On** checkbox on the corresponding Channel dialog, or touch-and-hold (right-click) on the descriptor box and choose **Off**.

From the front panel, press the **Channel button** (1-4) to turn on the trace; press again to turn it off.

NOTE: The default is to display each trace type in its own grid (e.g., Channels together, Zooms together, etc.). Use the Display menu to change how traces are arranged.

Other Traces

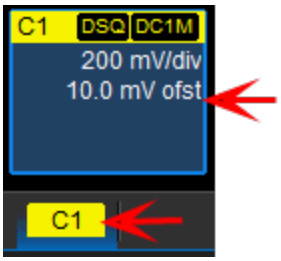
Quickly create zoom or math traces by touching the **Zoom** or **Math** action toolbar button.

Activate Trace

Although several traces may be open and appear on the grid, only one at a time is active. Touch the trace descriptor box to activate the trace. A highlighted descriptor box indicates the trace is active. All actions now apply to that trace until you activate another.

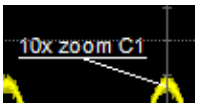
Active trace descriptor (left), inactive trace descriptor (right).

Whenever you activate a trace, the dialog at the bottom of the screen automatically switches to the appropriate setup dialog. The tab at the top of the dialog shows to which trace it belongs.



Channel descriptor label matches Channel dialog tab.

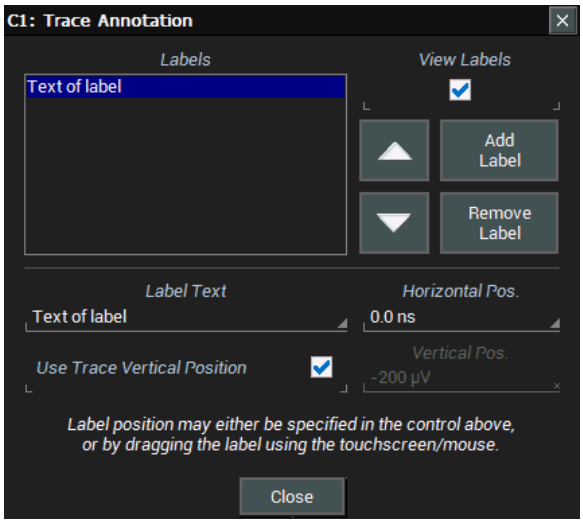
Annotating Traces



The Label function gives you the ability to add custom annotations to traces that are shown on the display. Labels are numbered sequentially in the order they were created. Once placed, labels can be moved to new positions or turned off.

Create Label

1. Touch the trace and choose **Set label...** from the context menu, or touch the trace descriptor box twice and touch the **Label toolbar button** on the 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. You can optionally check **Use Trace Vertical Position** instead of entering a Vertical Pos.

Reposition Label

Once placed, drag-and-drop labels to a new position on the grid, or reopen the Trace Annotation pop-up and enter a new **Horizontal Pos.** and **Vertical Pos.**

Edit/Remove Label

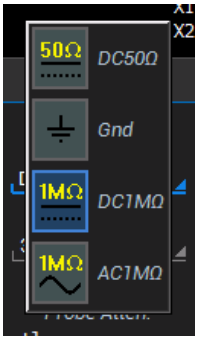
Open the Trace Annotation pop-up and select the **Label**. You can use the **Up/Down arrow** keys to scroll the list. Change the **Label Text** or **Horizontal** and **Vertical Pos.**(itions). Touch **Remove Label** to delete it.

Turn On/Off Labels

After labels have been placed, you can turn on/off all labels at once by opening the Trace Annotation dialog and selecting/deselecting the **View labels** checkbox.

Entering/Selecting Data

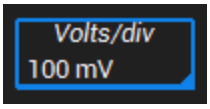
Touch



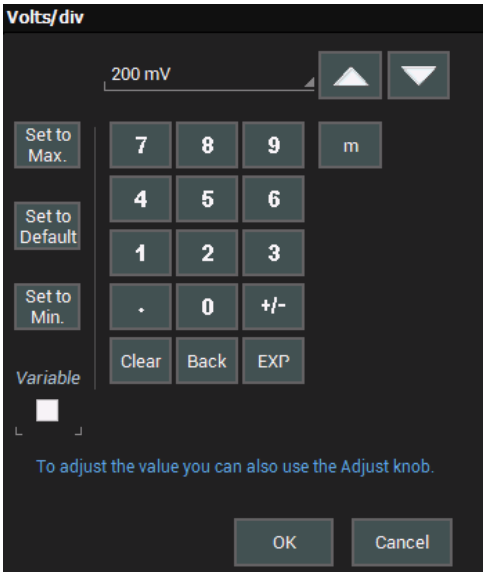
Touch once to activate a control. In some cases, you'll immediately see a pop-up menu of options. Touch one to select it.

TIP: You can touch the **Icon**  or **List**  buttons where they appear on larger pop-ups to change how menu options are displayed.

Touch & Type



In other cases, data entry fields appear highlighted in blue when you touch them. When a data entry field is highlighted, it is active and can be modified by using the front panel Adjust knob. Or, touch it again and use the pop-up menu or keypad to make an entry.



You'll see a pop-up keypad when you touch twice on a numerical data entry field. Use it exactly as you would a calculator. When you touch OK, the calculated value is entered in the field.

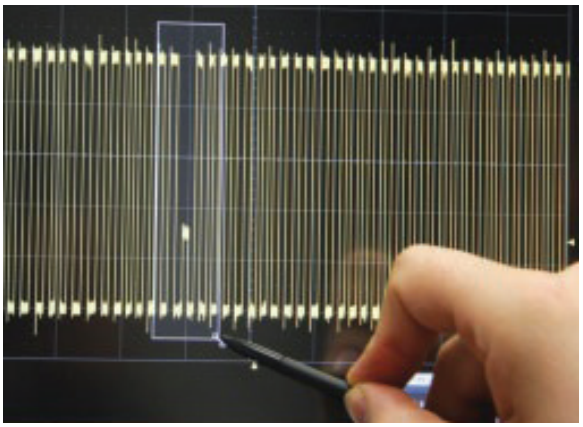
The Set to... buttons quickly enter the maximum, default or minimum value for that field.

The Up and Down arrow buttons increment/decrement the displayed value.

The Variable checkbox allows you to make fine increment changes when using the Up and Down arrow buttons.

Touch & Drag

Touch-and-drag cursor lines and annotation labels to reposition them on the grid; this is the same as setting the values on the dialog.



Touch-and-drag to draw a selection box around part of a trace to quickly zoom that portion.

Printing/Screen Capture

The Print function captures an image of the display and outputs it according to your Hardcopy settings.

There are three ways to print a capture of the screen:

- Touch the **front panel Print button**.
- Choose **File > Print**.
- Choose **Utilities > Utilities Setup > Hardcopy tab** and touch the **Print button** to the far right of the dialog.

NOTE: When the front panel Print button is configured to capture the screen as a LabNotebook entry, only the File and Utilities menu print options will function according to your Hardcopy setup.

Oscilloscope Application Window

The oscilloscope application runs on a Windows operating system and functions exactly as do other Windows applications.

To minimize the application window and show the Windows desktop, choose **File > Minimize**. To restore the window after minimizing, touch the oscilloscope display icon in the lower right corner of the desktop.

To exit the application window, choose **File > Exit**. When you exit the application, the oscilloscope operating system continues to run. To reload the application after exiting, touch the **Start DSO** desktop shortcut.

Language Selection

To change the language that appears on the touch screen:

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

To also change the language of the Windows operating system dialogs:

1. Choose File > Minimize to hide the oscilloscope display and show the Windows Desktop.
2. From the Windows task bar, choose **Start > Control Panel > Clock, Language and Region**.
3. Under Region and Language select **Change Display Language**.
4. Touch the **Install/Uninstall Languages** button.
5. Select **Install Language** and **Browse Computer or Network**.
6. Touch the **Browse** button, navigate to D:\Lang Packs\ and select the language you want to install. The available languages are: German, Spanish, French, Italian, and Japanese. Follow the installer prompts.

   NOTE: Other language packs are available from Microsoft's website.
7. Reboot the oscilloscope after changing the language.

Screen Saver

As on any Windows PC, a screen saver can be enabled to begin after a preset idle time, or disabled:

1. Minimize the oscilloscope application by choosing **File > Minimize** from the menu bar.
2. Open the Windows Control Panel to change Appearance and Personalization settings.
3. Touch the oscilloscope icon at the bottom right of the desktop to restore the instrument display.

Front Panel

Most front panel controls duplicate functionality available through the touch screen display and are described on the following pages.

Many knobs on the front panel function one way if turned and another if pushed like a button. When a knob is multi-plexed, 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.

Top Row Buttons

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

Clear Sweeps resets the acquisition counter and any cumulative measurements.

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

When you push the **Intensity** knob, the oscilloscope switches into WaveStream acquisition mode. The WaveStream indicator lights to show it is on.

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 sets Auto trigger mode, which triggers the oscilloscope after a time-out, even if the trigger conditions are not met.

Normal sets Normal trigger mode, which triggers the oscilloscope each time a signal is present that meets the conditions set for the type of trigger selected.

Single sets Single trigger mode, which arms the oscilloscope to trigger once (single-shot acquisition) when the input signal meets the trigger conditions set for the type of trigger selected. If the scope 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.

Measure, 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 **Measure** 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, while a readout of the values appears on the trace descriptor box. There are three 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 types until the desired type is found. The type "no cursors" turns off the cursor display.

Cursor knobs reposition the selected cursor line when turned. Each knob controls one line. Push the knob resets the cursor to the default position. When both Horizontal and Vertical cursors are displayed, each knob controls two lines, and pushing the knob switches the line that is being controlled.

Adjust

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 the knob is turned.

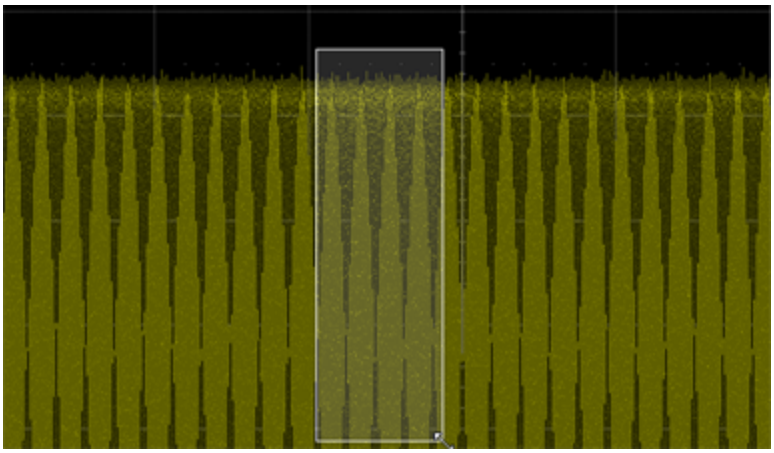
The **Touch Screen** button enables/disables the touch screen controls.

Zooming Waveforms

The Zoom function magnifies a selected region of a trace. On WaveSurfer 10 model oscilloscopes, you can display up to four zoom traces (Z1-Z4) taken from any channel, math, or memory trace.

Creating Zooms

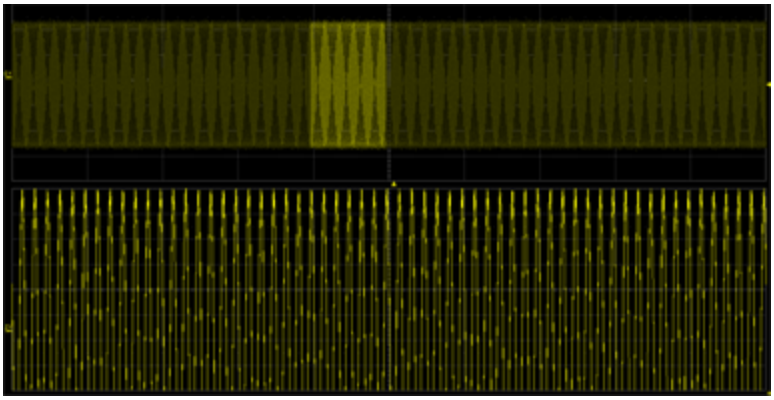
To create a zoom, touch -and-drag to draw a selection box around any part of the source waveform.



Selection box over trace.

The zoom will resize the selected portion 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.

The zoom opens in a new grid, with the zoomed portion of the source trace highlighted. New zooms are turned on and visible by default. However, you can turn off a particular zoom if the display becomes too crowded, and the zoom settings are saved in its Zx location, ready to be turned on again when desired.



Adjust Zoom

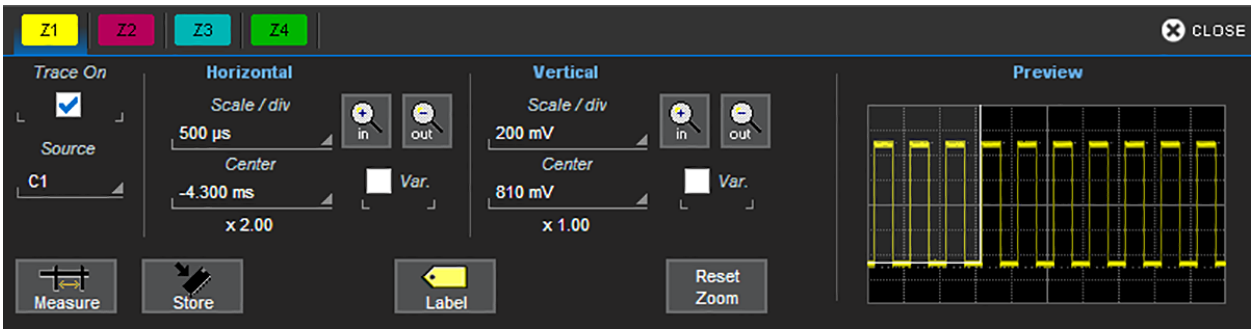
The zoom's Horizontal units will differ from the signal timebase because the zoom is showing a calculated scale, not a measured level. This allows you to adjust the zoom factor using the front panel knobs or the Zoom dialog controls however you like without affecting the timebase (a characteristic shared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

Turn off Zoom

To close the zoom, either touch the zoom descriptor box twice to open the Zoom dialog and deselect **Trace On**, or touch the zoom trace to open the context menu and choose **Off**.

Zoom Controls

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



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. They are only displayed on WaveSurfer 10 oscilloscopes with the WS10-ADT option installed.

Zoom Factor Controls

These controls on the **Z*x* dialogs** appear throughout the oscilloscope software:

- **Out** and **In buttons** increase or decrease the magnification of the zoom, and consequently change the Horizontal and Vertical Scale settings. Continue to touch either button until you've achieved the desired level of zoom.

- **Var. checkbox** enables variable zooming in increments finer than the default 1, 2, 5, 10 step increments. When checked, each touch of the zoom control buttons changes the degree of magnification by a single increment.
- **Horizontal Scale/div** sets the amount of time represented by each horizontal division of the grid. It is the equivalent of Time/div, only unlike the Timebase setting, it may be set differently for each zoom, math function, or memory trace.
- **Vertical Scale/div** sets the voltage level represented by each vertical division of the grid; it's the equivalent of V/div used for channel settings.
- **Horizontal/Vertical Center** sets the voltage or time that is to be at the center of the screen on the zoom trace. The horizontal center is the same for all zoom traces.
- **Reset Zoom** returns the zoom to x1 magnification.

Vertical

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

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 Channel descriptor box always shows the current Vertical Scale setting.

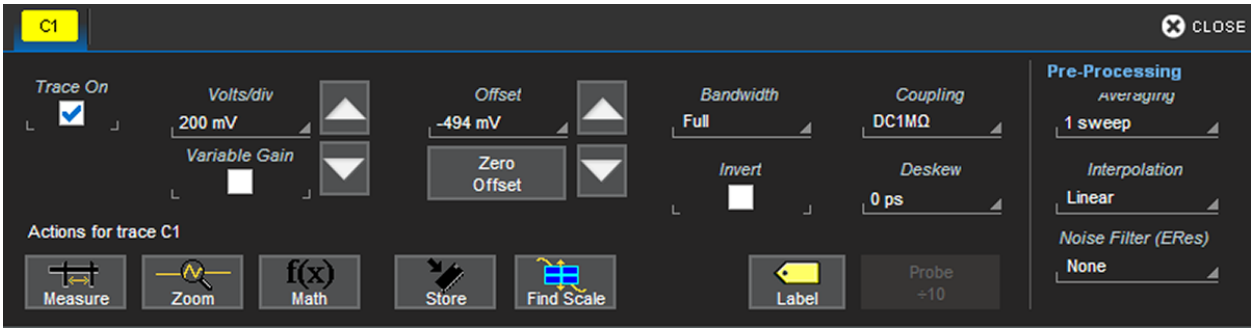
Vertical settings are made on the Channel dialog, labeled **C*x*** after the corresponding channel. To access the Channel dialog, choose **Vertical > Channel <#> Setup** from the menu bar, or touch the **Channel descriptor box**.

The C*x* dialog contains:

- Channel Settings for scale, offset, coupling, bandwidth, and probe attenuation.
- Pre-Processing Settings to set up pre-acquisition processes that will affect the waveform, such as noise filtering and interpolation.

If a Teledyne LeCroy probe is connected to the channel, a Probe dialog appears behind the C*x* dialog.

Channel Settings



Volts/div sets the vertical scale (aka gain or sensitivity). Select **Variable Gain** adjustment or leave the checkbox clear for fixed adjustment.

Offset adds a defined value of DC offset to the signal as acquired by the input channel. This may helpful in order to display a signal on the oscilloscope 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 at a variety of fixed settings. The exact settings vary by model. To limit bandwidth, select a filter from this field.

Invert inverts the waveform for the selected channel.

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

CAUTION. The maximum input voltage depends on the input used. Limits are displayed on the front of the oscilloscope. Whenever the voltage exceeds this limit, the coupling mode automatically

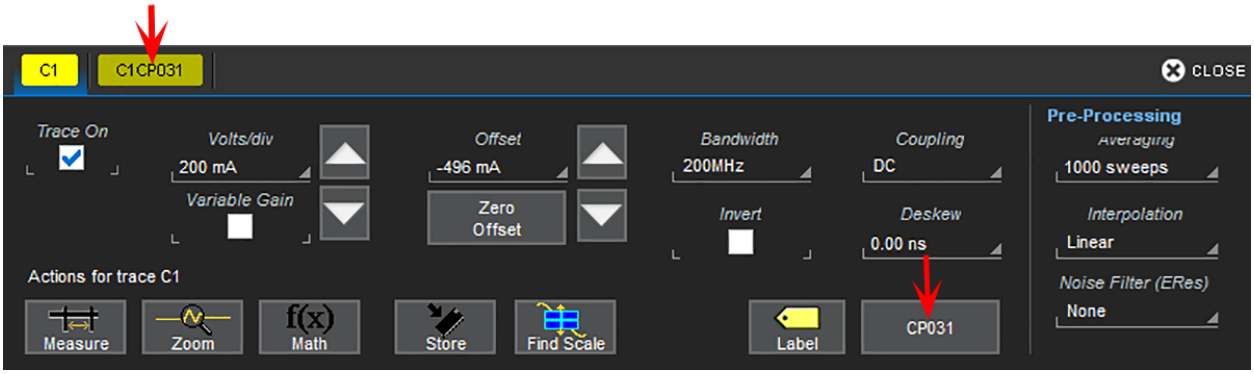
switches to GROUND. You then have to manually reset the coupling to its previous state. While the unit does provide this protection, damage can still occur if extreme voltages are applied.

Deskew adjusts the amount of horizontal time offset to compensate for propagation delays caused by different probes or cable lengths. The valid range depends on the current timebase setting. The Math deskew function performs the same activity.

Probe Settings

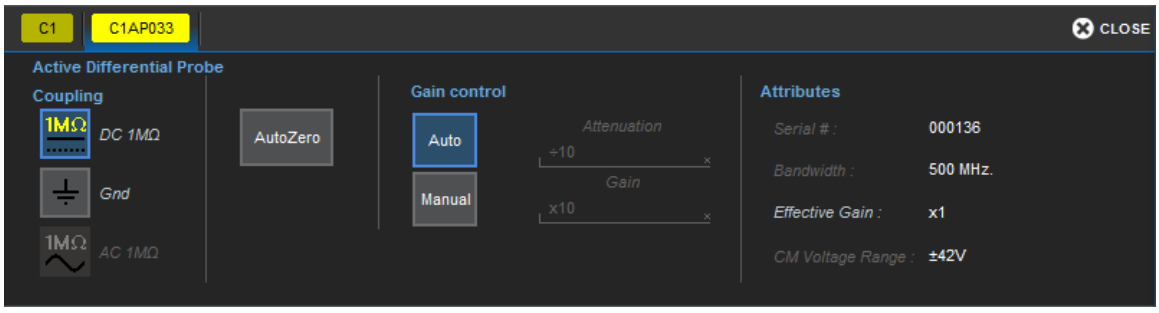
When a Teledyne LeCroy-compatible probe is connected to the oscilloscope input, the probe is automatically identified and the model name displayed on the Channel dialog under the "Probe" heading. Also, the Probe dialog bearing the probe name is added to the right of the Channel dialog. When a probe is not connected, the Channel dialog shows only the C*x* tab for vertical setup.

When third-party probes are connected, an **Attenuation** field appears on the C*x* dialog, with a default value of /1, allowing you to enter attenuation and rescale values manually.



Channel dialog with tab for connected probe.

The Probe Dialog displays probe attributes and (depending on the probe type) allows you to AutoZero or DeGauss probes from the oscilloscope touch screen. Other settings may appear, as well, depending on the probe model.



Probe dialog showing the connected probe's control attributes.

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.

CAUTION. Remove the probe from the circuit under test before initializing Auto Zero.

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.

CAUTION. Remove the probe from the circuit under test before initializing DeGauss.

Auto Setup

Auto Setup quickly configures the essential oscilloscope settings based on the first input signal it finds, starting with Channel 1. If nothing is connected to Channel 1, it searches Channel 2 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:

1. Either press the **Auto Setup button** on the front panel, or choose **Auto Setup** from the Vertical, Timebase, or Trigger menus. All these options perform the same function.
2. Press the Auto Setup button again or use the touch screen display to confirm Auto Setup.

Restore Default Setup

Restore the oscilloscope to its factory default state by pressing the front panel **Default Setup** button. You can also restore default settings by choosing **File > Recall Setup > Recall Default**.

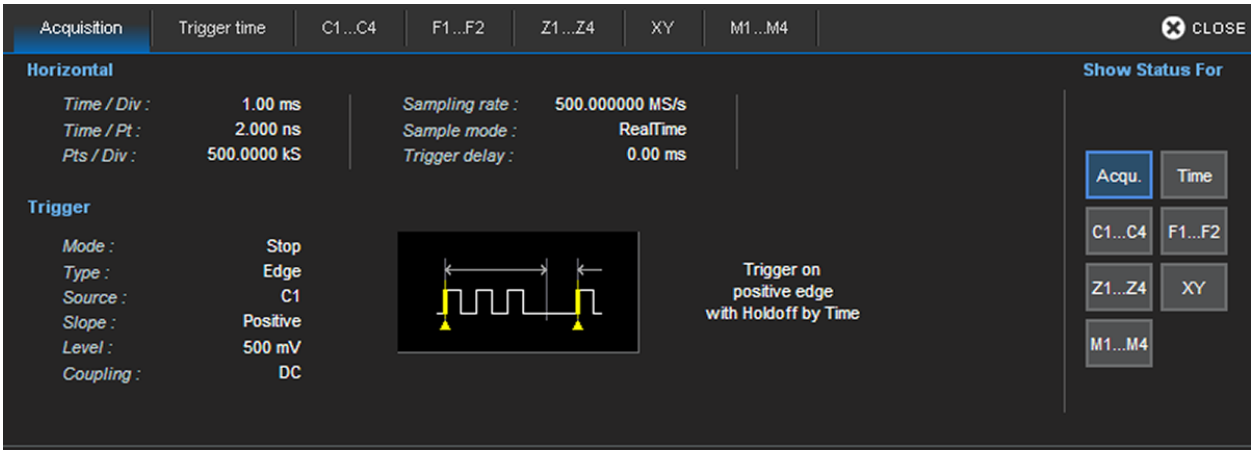
Default settings for your oscilloscope include the following:

Channel/Vertical	C1-C2 on at 50 mV/div Scale, 0 V Offset
Timebase	Real Time Sampling at 50 ns/div, 0 Delay, 2.0 kS at 4 GS/s, 100 kS Memory
Trigger	C1 with an Auto Positive Edge, DC Coupling, 0 V Level
Display	Auto Grid
Cursors	Off
Measurements	Cleared
Math	Cleared

Viewing Status

All oscilloscope 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, Math menus (e.g., Channel Status, Acquisition Status).



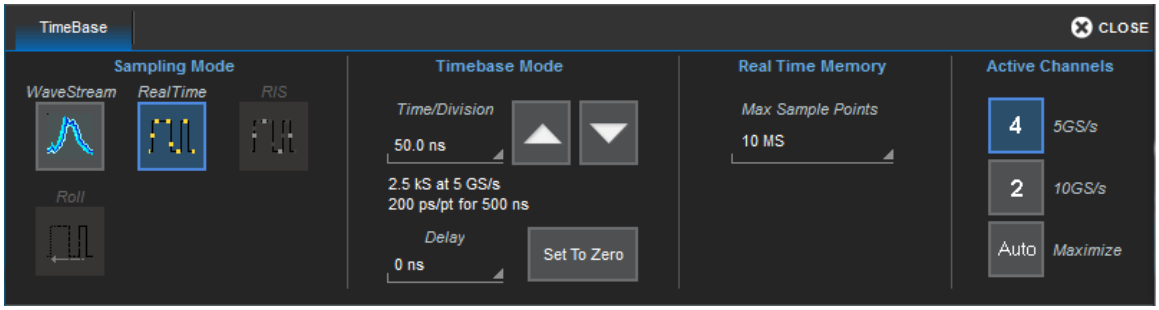
Timebase

Timebase, also known as Horizontal, settings control the trace 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, including sampling mode selection, is done on the Timebase dialog, which can be accessed by either choosing **Timebase > Horizontal Setup** from the menu bar, or touching the **Timebase descriptor box**.

The Timebase dialog contains settings for Sampling Mode, Timebase Mode, Real Time Memory, and Active Channels.

Timebase Settings



Sampling Mode

Choose fromWaveStream, Real Time, Sequence,RIS, orRoll mode.

NOTE: Sequence mode available only on oscilloscopes with the WS10-ADT option installed.

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.

Delay is the amount of time relative to the trigger event to display on the grid. In Real Time sampling mode, the trigger event is placed at time zero on the grid. Delay may be time pre-trigger, entered as a negative value, or post-trigger, entered as a positive value. Raising/lowering the Delay value has the effect of shifting the trace to the right/left, enabling you to focus on the relevant portion of longer acquisitions.

Set to Zero returns Delay to zero.

Real Time Memory

Max. Sample 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.

Active Channels

These settings enable you to control the distribution of memory to achieve longer acquisitions through a single channel if needed.

4 (or 2 in 2 channel scopes) utilizes the per channel maximum memory (10 Mpts/ch standard, 16 Mpts/ch with the WS10-ADT option).

2 (or 1 in 2 channel scopes) distributes the total maximum memory across only two channels (20 Mpts/ch standard, 32 Mpts/ch with the WS10-ADT option) . Channels 2 and 3 (in 4 channel scopes) and Channel 2 (in 2 channel scopes) are available for use.

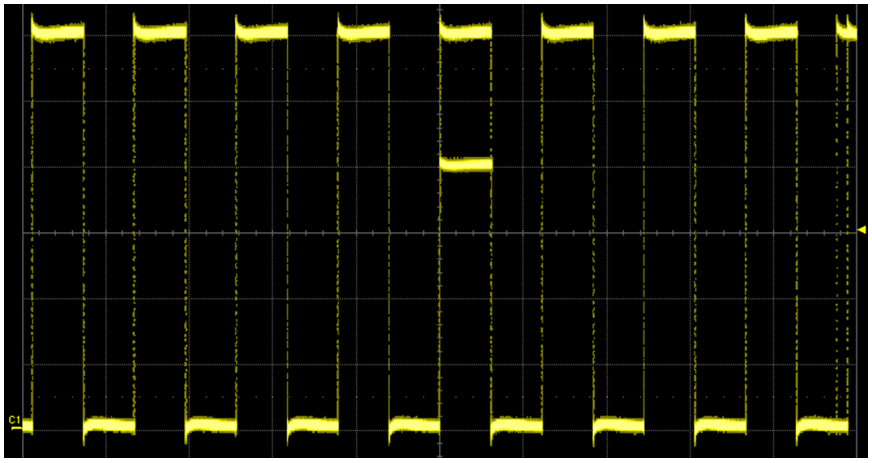
Auto allows the scope to make this decision based on which channels are currently in use.

Sampling Modes

WaveStream Sampling Mode

WaveStream provides a vibrant, intensity graded display with a fast update rate to closely simulate the look and feel of an analog oscilloscope. WaveStream is most helpful in viewing signals that have signal jitter or signal anomalies, or for applying a visual check before creating an advanced trigger or WaveScan setup to locate an unusual event.

WaveStream mode operates at up to 80 GS/s with an update rate of up to several thousand waveforms/second for better capture of higher frequency abnormal events. Time/div must be set to 50 ms or faster to use WaveStream.



Waveform captured in WaveStream sampling mode, showing hard-to-find runt.

To use WaveStream, select it as the Sampling Mode when making other settings on the Timebase dialog, or press the front panel **Intensity knob**. The WaveStream (ACQ) indicator next to the knob will light to show you are now in WaveStream mode. Press the knob again to exit WaveStream Mode.

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, the waveform is horizontally positioned so that the trigger event is time zero on the grid.

The relationship between sample rate, memory, and time can be expressed as:

Capture Interval = 1/Sample Rate X Memory
Capture Interval/10 = Time Per Division

In Real Time sampling mode, the acquisition can be displayed for a specific period of time (or number of samples) either before or after the trigger event occurs, known as trigger delay. This allows you to isolate and display a time/event of interest that occurs before or after the trigger event.

- **Pre-trigger delay** displays the time prior to the trigger event. This can be set from a time well before the trigger event to the moment the event occurs, up to the oscilloscope's maximum sample record length. How much actual time this represents depends on your timebase setting. When set to the maximum allowed pre-trigger delay, the trigger position (and zero point) is off the grid (indicated by the trigger delay arrow at the lower right corner), and everything you see represents pre-trigger time.
- **Post-trigger delay** displays time following the trigger event. Post-trigger delay can cover a much greater lapse of time than pre-trigger delay, up to the equivalent of 10,000 time divisions after the trigger event occurred. When set to the maximum allowed post-trigger delay, the trigger point may actually be off the grid far to the left of the time displayed.

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.

Sequence Sampling Mode

Sequence sampling mode is available on WaveSurfer 10 oscilloscopes with the WS10-ADT option installed.

In Sequence Mode, the complete waveform consists of a number of fixed-size segments (see the instrument specifications at teledynelecroy.com for the limits). The oscilloscope uses the sequence timebase setting to determine the capture duration of each segment as 10 x time/div. With this setting, the oscilloscope uses the desired number of segments, maximum segment length, and total available memory 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. You can also make time measurements between events on selected segments using the full precision of the acquisition timebase.

Set Up Sequence Mode

When setting up Sequence Mode, you define the number of fixed-size segments acquired in single-shot mode (see the instrument specifications for the limits). The oscilloscope uses the sequence timebase setting to determine the capture duration of each segment. Along with this setting, the oscilloscope uses the number of segments, maximum segment length, and total available memory to determine the actual number of samples or segments, and time or points.

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

   NOTE: While optional, Timeout ensures that the acquisition will complete in a reasonable amount of time and control of the oscilloscope will return to the operator/controller without having to manually stop the acquisition.

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

   NOTE: Once acquisition has started, you can interrupt it at any time by pressing the **Stop** front panel button. In this case, the segments already acquired will be retained in memory.

View Segments In Sequence Mode

When in Sequence Mode, you can view individual segments easily using the **Zoom dialog**. The Zoom trace defaults to Segment 1. You can move to later segments by changing the values in **First** segment to display and **Num**(ber) of segments to display at once.

TIP: By changing the Num field value to 1, you can use the front panel Adjust knob to scroll through each segment in order.

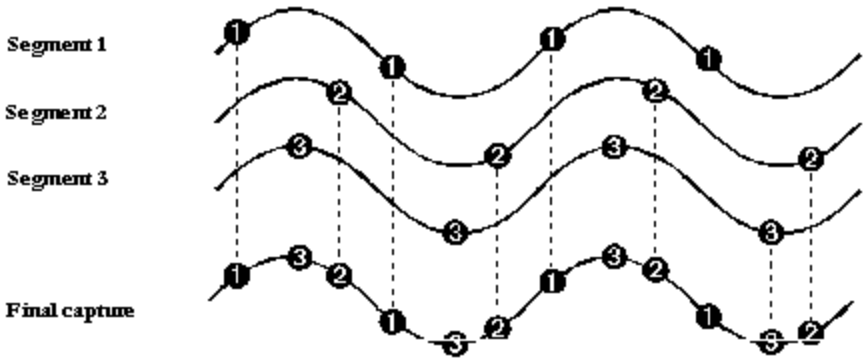
Channel descriptor boxes indicate the total number of segments acquired. Zoom descriptor boxes show the . As with all other Zoom traces, the zoomed segments are highlighted on the source trace.

Use the to change the scale factors of the trace.

RIS Sampling Mode

RIS (Random Interleaved Sampling) allows effective sampling rates higher than the maximum single-shot sampling rate. It is used on repetitive waveforms with a stable trigger. 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.

The instrument requires multiple triggers to complete an acquisition. The number depends on the sample rate: the higher the sample rate, the more triggers are required. It then interleaves these segments (as shown in the following illustration) 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.



Interleaving of sample in RIS sampling mode.

Roll 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 connected to each channel are updated every time the roll mode buffer is updated, as if new data is available. This resets statistics on every step of Roll mode that is valid because of new data.

Timebase must be set to 100 ms/div or slower to enable Roll mode selection. Roll mode samples at ≤ 5 MS/s.

NOTE: If the processing time is greater than the acquire time, the data in memory is overwritten. In this case, the instrument issues the warning, "Channel data is not continuous in ROLL mode!!!" and rolling starts again.

History Mode

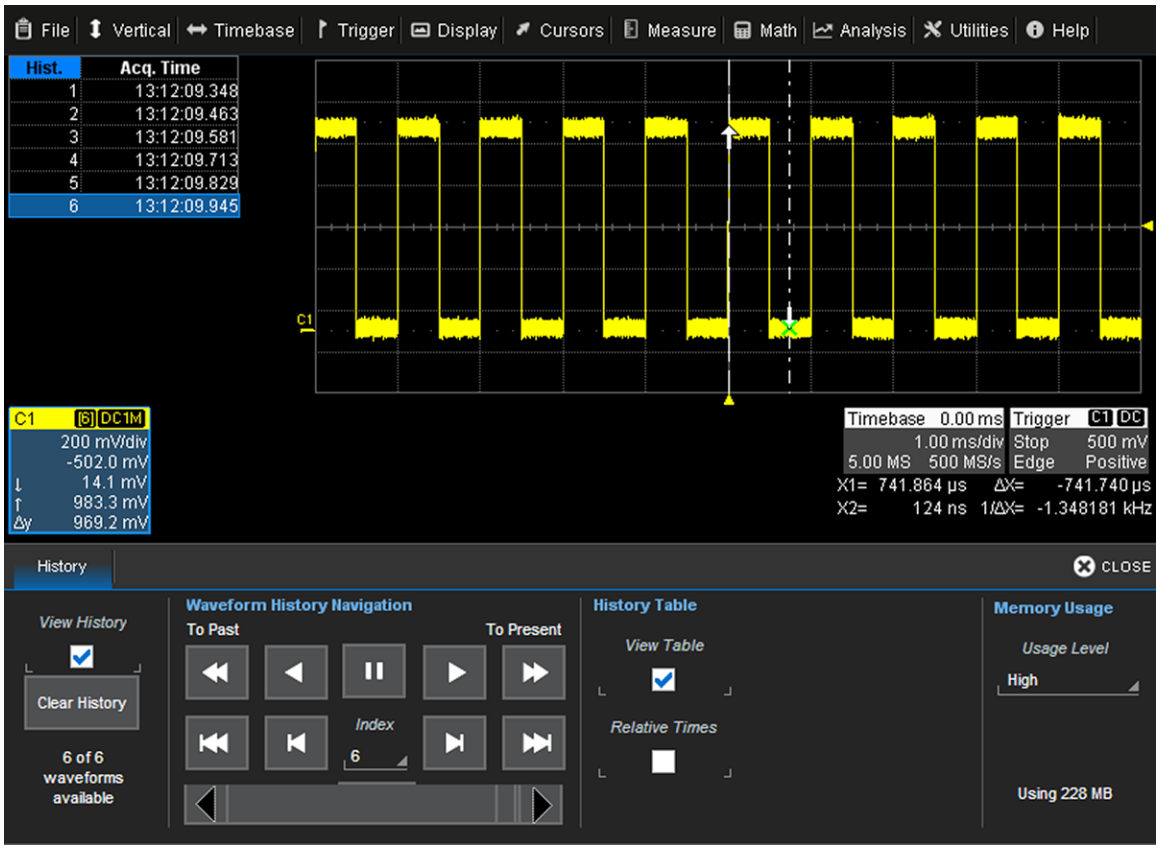
History Mode is available on WaveSurfer 10 oscilloscopes with the WS10-ADT option installed.

History Mode allows you to review any acquisition saved in the oscilloscope's 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 size of the oscilloscope memory.

To view history:

1. Choose Timebase > History Mode.
2. Press the front panel **History Mode button**, or choose **Timebase > History Mode**.
3. Select **View History** to enable the history display, and **View Table** to display the index of records. Optionally, select to show **Relative Times** on the table.



4. Choose a single acquisition to view by entering its **Index** number on the dialog or selecting it from the table of acquisitions.

   OR

   Use the Navigation buttons to "scroll" the history of acquisitions.

   - The top row of buttons scrolls continuously and are (left to right): Fast Backward, Slow Backward, Pause, Slow Forward, Fast Forward.
   - The bottom row of buttons steps one record at a time and are (left to right): Back to Start, Back One, Go to Index (#), Forward One, Forward to End.

5. Entering History Mode automatically stops new acquisitions. To leave History Mode, restart acquisition by pressing one of the front panel Trigger Mode buttons.

Trigger

While the oscilloscope is continuously sampling signal when it is turned on, it can only display up to its maximum memory in data samples. Triggers select an exact event/time in the waveform to display on the oscilloscope screen so that memory is not wasted on insignificant periods of the signal. For all trigger types, you can set:

- Pre-trigger or post-trigger delay—time relative to the trigger event displayed on screen (although the trigger itself may not be visible).
- Time between sweeps—how often the display is refreshed.

Unless modified by a pre- or post-trigger delay, the trigger event occurs at point zero at the center of the grid, and an equal period of time before and after this point is shown to the left and right of it.

In addition to the trigger type, the trigger mode determines how the oscilloscope behaves in the presence or absence of a trigger event.

Trigger capabilities include:

- **Simple Triggers** activated by basic waveform features such as an edge with a positive or negative slope or width.
- **Pattern Triggers** that fire when a pattern condition occurs on selected input channels.
- **SMART Triggers**, sophisticated triggers that enable you to create basic or complex trigger conditions. Use SMART Triggers for signals with rare features, like glitches.

Trigger Modes

The trigger mode determines how the oscilloscope sweeps, or refreshes, the display. This can be set from the Trigger menu or from the front panel Trigger control group.

Auto mode causes the oscilloscope to sweep without a set trigger. An internal timer triggers the sweep after a preset timeout period so that the display refreshes continuously. Otherwise, Auto functions the same as Normal when a trigger condition is found.

In **Normal** mode, the oscilloscope sweeps only if the input signal reaches the set trigger point. Otherwise it continues to display the last acquired waveform.

In **Single** mode, one sweep occurs each time you choose **Trigger > Single** or press the front panel **Single** button.

Stop pauses sweeps until you select one of the other three modes.

Trigger Types

These are the trigger types available for selection. If the trigger is part of a subgroup (e.g., Smart), first choose the subgroup from among the basic types to display all the trigger options.

Basic Triggers

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 high and low voltage levels on analog channels, or just a digital logic pattern omitting any analog inputs. Likewise, if your oscilloscope does not have digital input capability, the pattern can be set using voltage levels on analog channels alone. You can stipulate the voltage level/logic threshold for each analog or digital input independently.

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

Qualified arms the trigger on the A event, then fires on the B event. In Normal trigger mode, it automatically resets after the B event. The A event can be an Edge, State, Pattern, or PatState (a pattern that persists over a user-defined number of events or time). The options for the B event depend on the type of A event. If A is a digital Pattern or PatState, B can only be an Edge.

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

Smart Triggers

The Smart subgroup triggers allow you to apply Boolean logic conditions to the basic signal characteristics of level, slope, and polarity to determine when to fire the trigger.

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.

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

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-shot applications with a 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.

Serial Triggers

The **Serial** trigger type will appear if you have installed protocol-specific serial data trigger and decode options. Select this type to open the serial trigger setup dialogs. Instructions for using all serial data options are available from our website at teledynelecroy.com/serialdata.

Setting Up Triggers

To access the Trigger setup dialogs, do one of the following:

- Choose **Trigger > Trigger Setup** from the menu bar
- Press the **front panel Trigger Setup** button
- Touch the **Trigger descriptor box**

The main Trigger dialog contains the trigger type selections. On oscilloscopes with the Mixed Signal option, many trigger types can be set on either analog channels, including the External Trigger input, or digital lines. For digital triggering instructions, see the Operator's Manual for your Mixed Signal accessory.

Other controls will appear depending on the trigger type selection (e.g., Slope for Edge triggers). These are described in the set up procedures for each trigger.

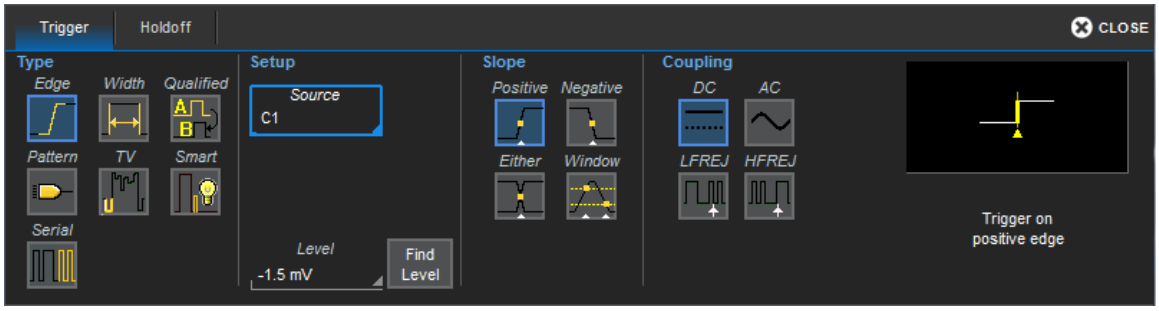
The trigger condition is summarized in a preview window at the far right of the Trigger dialog. Refer to this to confirm your selections are producing the trigger you want.

Edge Trigger

Edge triggers upon a achieving a certain voltage level in the positive or negative slope of the waveform. It is the default trigger selection on standard oscilloscopes.

NOTE: Alternatively, you may choose a Slope of **Window** and enter the **Upper Level** and **Lower Level** voltage that define the window. The trigger fires when the signal leaves the widow.

On the Trigger dialog, select **Edge** trigger type to display the controls.



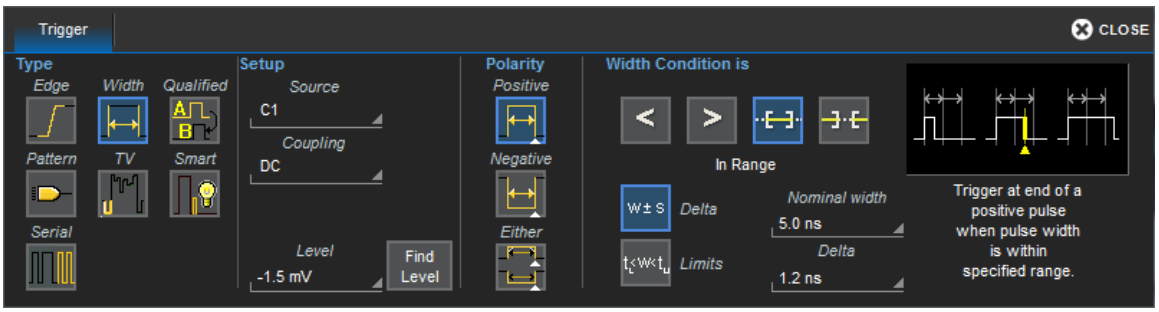
1. Choose the **Source** signal input.
2. Enter the voltage **Level** upon which to trigger.

   The **Find Level** button sets the Level to the signal mean.
3. Choose the **Slope** (edge) of the wave on which to trigger.
4. Choose the type of signal **Coupling** at the input. Choices are:
   - **DC** - All the signal's 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** - The signal is capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
   - **LFREJ** - The signal is 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** - Signals are DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz (used for triggering on low frequencies).

Width Trigger

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

On the Trigger dialog, select **Width** trigger type to display the controls.

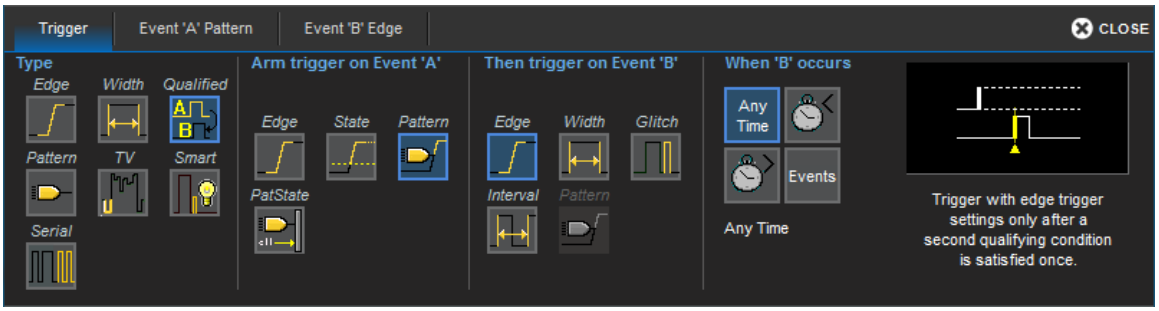


1. Choose the **Source** input.
2. Choose the type of signal **Coupling** at the input. Choices are:
    - **DC** - All the signal's frequency components are coupled to the trigger circuit for high frequency bursts or where the use of AC coupling would shift the effective trigger level.
    - **AC** - The signal is capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
    - **LFREJ** - The signal is coupled through a capacitive high-pass filter network, DC is rejected and signal frequencies below 50 kHz are attenuated. Best used for stable triggering on medium to high frequency signals.
    - **HFREJ** - Signals are DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz. Best used for triggering on low frequencies.
3. Choose the **Polarity** at which to measure pulse width.
4. Enter the voltage **Level** at which to measure pulse width. The Find Level button sets the level to the signal mean.
5. Use **Width Condition is** settings to create an expression describing the triggering pulse width. This may be:
    - Any width **Less Than** an **Upper Value**.
    - Any width **Greater Than** a **Lower Value**.
    - Any width **In Range** or **Out Range** of values. You may describe the range using either:
        - **Limits**, an absolute **Upper Value** and **Lower Value**.
        - **Delta**, any **Nominal width** plus or minus a **Delta** width.

Qualified Trigger

Qualified arms the trigger on the A event, then fires on the B event. In Normal trigger mode, it automatically resets after the B event. The options for the B event depend on the type of A event. You may apply additional Holdoff by time or number of events.

On the Trigger dialog, select **Qualified** trigger type to display the controls.



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

- State, any voltage measured above or below a threshold Level.
- PatState, a pattern that persists over a user-defined number of events or time. Like Pattern triggers, PatState events may be analog voltage patterns, digital logic patterns, or a mix of both, depending on the oscilloscope's capabilities.

NOTE: On a standard oscilloscope, Pattern and PatState events will default to the analog pattern setup dialog. On a Mixed-Signal oscilloscope, Pattern and PatState events will default to the digital pattern setup dialog.

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

Pattern Trigger

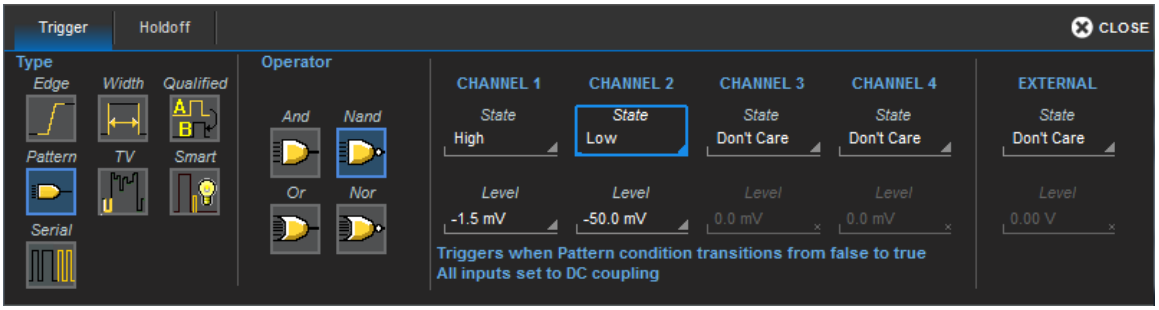
Pattern is the default trigger when the Mixed Signal option is connected to the oscilloscope, as these users generally wish to find and trigger upon digital logic patterns.

However, a Pattern trigger can also be set on a user-defined pattern of High or Low voltage levels in analog channels (including the External Trigger input), or a combination of digital and analog patterns when Mixed Signal capabilities are available.

See the *MS-250 Operator's Manual* or *MS-500 Operator's Manual* delivered with your Mixed Signal option for instructions on setting up a digital pattern trigger.

To set up an analog pattern trigger, on the Trigger dialog, select **Pattern** trigger type.

The standard dialog for setting up an analog Pattern trigger includes all the controls for setting the pattern and the voltage threshold on the same dialog.

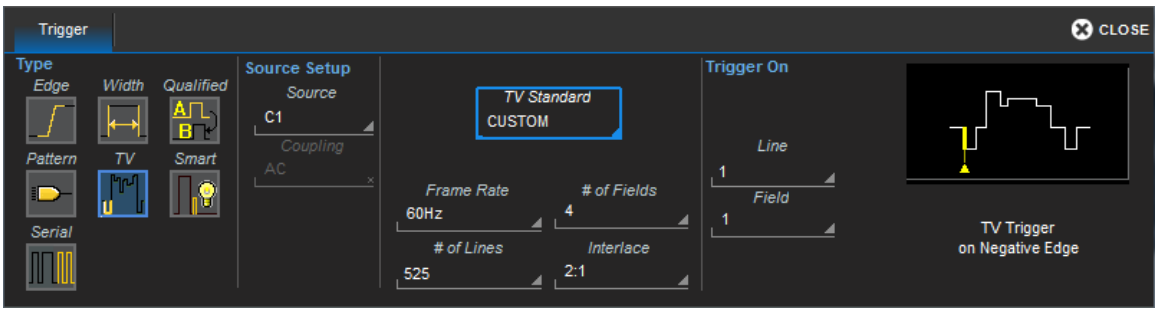


1. Select the Boolean **Operator** (AND, NAND, OR, or NOR) that describes the relationship among analog inputs (e.g., C1 must be High NAND C2 must be Low).
2. For each input to be included in the trigger pattern, and select what **State** it must be in (High, Low, or Don't Care) compared to the threshold Level you will set. Leave "Don't Care" selected for any input you wish to exclude.
3. For each input included in the trigger, enter the voltage threshold **Level**.
4. If you've included EXTERNAL as an input, open the **Ext tab** and enter the **Attenuation**.

TV Trigger

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

On the Trigger dialog, select **TV** trigger type to display the controls.

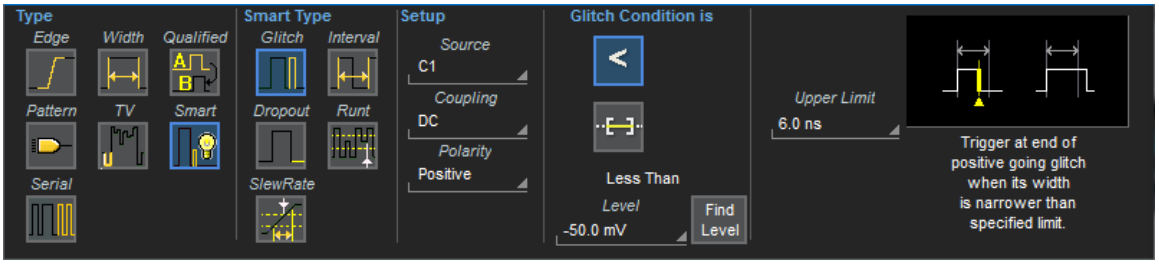


1. Choose the **Source** signal input.
2. Choose the signal **TV Standard**. To use a custom signal, also enter the:
    - **Frame Rate**
    - **# of Fields** per line
    - **# of Lines**
    - **Interlace** ratio
3. Choose the **Line** and **Field** upon which to trigger.

Glitch Trigger

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

On the Trigger dialog, select **Smart** trigger type, then **Glitch** to display the controls.

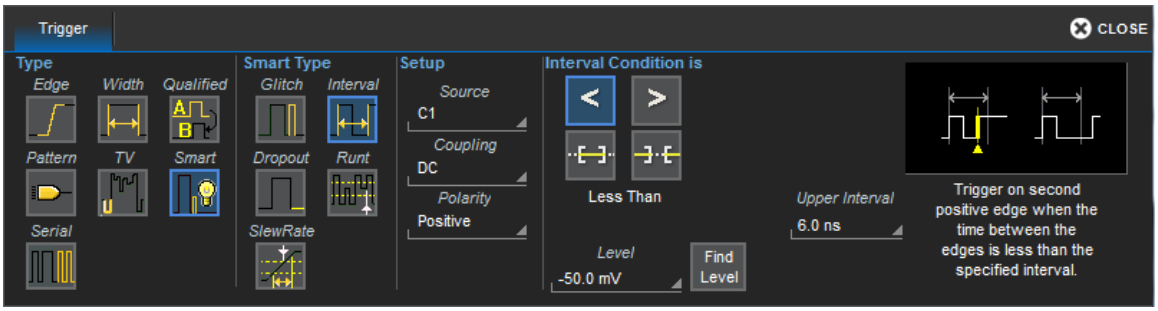


1. Choose the **Source** signal input.
2. Choose the type of signal **Coupling** at the input. Choices are:
   - **DC** - All the signal's frequency components are coupled to the trigger circuit for high frequency bursts or where the use of AC coupling would shift the effective trigger level.
   - **AC** - The signal is capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
   - **LFREJ** - The signal is 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** - Signals are DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz (used for triggering on low frequencies).
3. Choose the **Polarity** on which to trigger.
4. Enter the voltage **Level** at which to measure. The **Find Level** button sets the Level to the signal mean.
5. Use **Glitch Condition is** settings to create an expression describing the glitch width. This may be:
   - Any width **Less Than** an **Upper Value**.
   - Any width **In Range** of values marked by the specified **Upper Value** and **Lower Value**.

Interval Trigger

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.

On the Trigger dialog, select **Smart** trigger type, then **Interval** to display the controls.

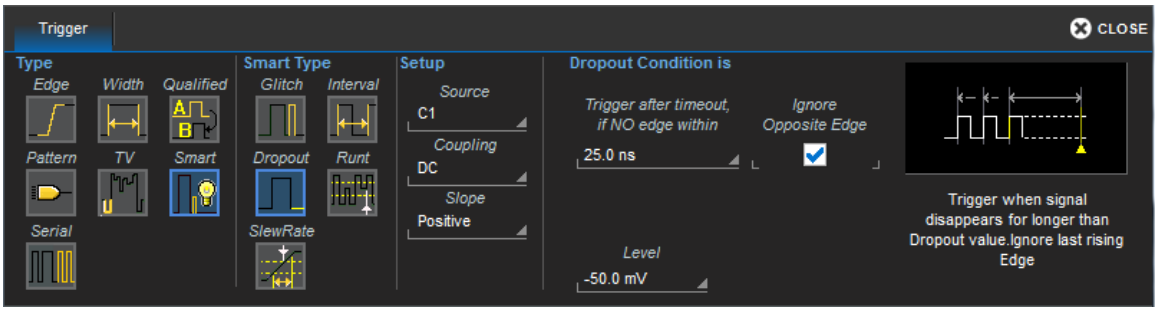


1. Choose the **Source** input.
2. Choose the type of signal **Coupling** at the input. Choices are:
    - **DC** - All the signal's frequency components are coupled to the trigger circuit for high frequency bursts or where the use of AC coupling would shift the effective trigger level.
    - **AC** - The signal is capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
    - **LFREJ** - The signal is 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** - Signals are DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz (used for triggering on low frequencies).
3. Choose the **Slope** (edge) from which to measure.
4. Enter the voltage **Level** at which to measure interval width. Where available, the Find Level button sets the level to the signal mean.
5. Use **Interval Condition is** settings to create an expression describing the triggering interval. This may be:
    - Any width **Less Than** an **Upper Value**.
    - Any width **Greater Than** a **Lower Value**.
    - Any width **In Range** or **Out Range** of values. You may describe the range using either:
        - **Limits**, an absolute **Upper Value** and **Lower Value**.
        - **Delta**, any **Nominal width** plus or minus a **Delta** width.

Dropout Trigger

Dropout triggers when a signal loss is detected. The trigger is generated at the end of the timeout period following the last edge transition that meets the trigger conditions. It is used primarily in single-shot applications with a pre-trigger delay.

On the Trigger dialog, select **Smart** trigger type, then **Dropout** to display the controls.

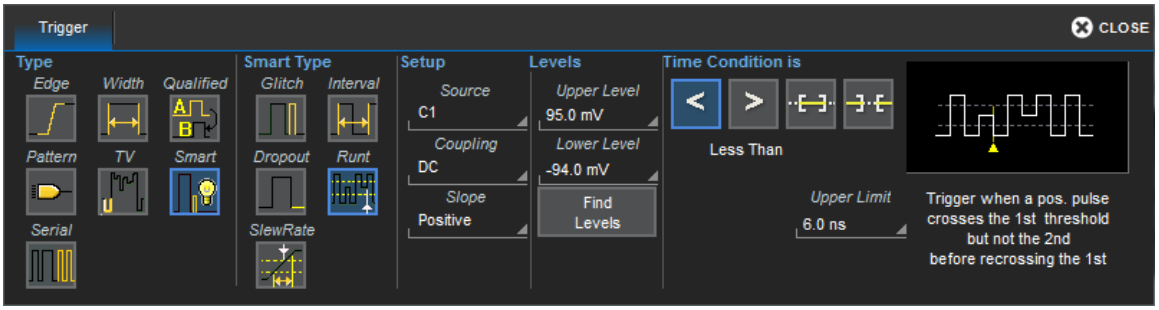


1. Choose the **Source** signal input.
2. Choose the type of signal **Coupling** at the input. Choices are:
   - **DC** - All the signal's frequency components are coupled to the trigger circuit for high frequency bursts or where the use of AC coupling would shift the effective trigger level.
   - **AC** - The signal is capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
   - **LFREJ** - The signal is 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** - Signals are DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz (used for triggering on low frequencies).
3. Choose the **Slope** (edge) and enter the voltage **Level** to watch for transitions. Where available, the **Find Level** button sets the Level to the signal mean.
4. Under **Dropout Condition is...**, enter the time interval after which to trigger if no transition occurs at that Slope and Level.

Runt Trigger

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 polarity and runt interval (width).

On the Trigger dialog, select **Smart** trigger type, then choose **Runt** to display the controls.

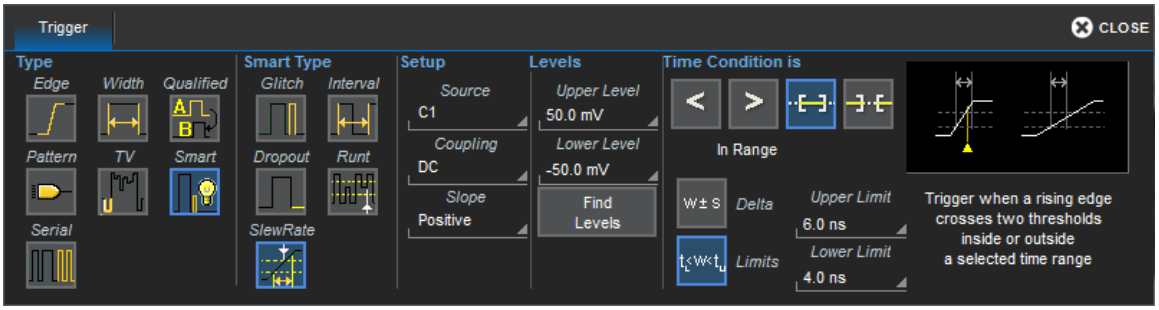


1. Choose the **Source** input.
2. Choose the type of signal **Coupling** at the input. Choices are:
    - **DC** - All the signal's frequency components are coupled to the trigger circuit for high frequency bursts or where the use of AC coupling would shift the effective trigger level.
    - **AC** - The signal is capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
    - **LFREJ** - The signal is 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** - Signals are DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz (used for triggering on low frequencies).
3. Choose the **Polarity** on which to measure.
4. Enter the voltage crossing **Upper Level** and **Lower Level**. Where available, the Find Level button sets the levels to the positive and negative signal mean.
5. Use **Time Condition is** settings to create an expression describing the runt interval (width). This condition is in addition to (AND) the voltage crossing levels. The interval may be:
    - Any width **Less Than** an **Upper Interval**.
    - Any width **Greater Than** a **Lower Interval**.
    - Any width **In Range** or **Out Range** of values. You may describe the range using either:
        - **Limits**, an absolute **Upper Interval** and **Lower Interval**.
        - **Delta**, any **Nominal width** plus or minus a **Delta** width.

SlewRate Trigger

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.

On the Trigger dialog, select **Smart** trigger type, then **Slew Rate** to display the controls.



1. Choose the **Source** input.
2. Choose the type of signal **Coupling** at the input. Choices are:
    - **DC** - All the signal's 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** - The signal is capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
    - **LFREJ** - The signal is 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** - Signals are DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz (used for triggering on low frequencies).
3. Choose the **Slope** (edge) from which to measure.
4. Enter the voltage crossing **Upper Level** and **Lower Level**. Where available, the Find Level button sets the level to the positive and negative signal mean.
5. Use **Time Condition is** settings to create an expression describing the interval within which both levels must be crossed. This may be:
    - Any time **Less Than** an **Upper Value**.
    - Any time **Greater Than** a **Lower Value**.
    - Any time **In Range** or **Out Range** of values. You may describe the range using either:
        - **Limits**, an absolute **Upper Value** and **Lower Value**.
        - **Delta**, any **Nominal width** plus or minus a **Delta** width.

Trigger Holdoff

The trigger holdoff function is available on WaveSurfer 10 oscilloscopes with the WS10-ADT option installed.

Holdoff is an additional condition that may be set for Edge and Pattern triggers. It can be expressed either as a period of time or an event count. Holdoff disables the trigger temporarily, even if the trigger conditions are met, until the holdoff conditions are also met. The trigger fires when the holdoff has elapsed.

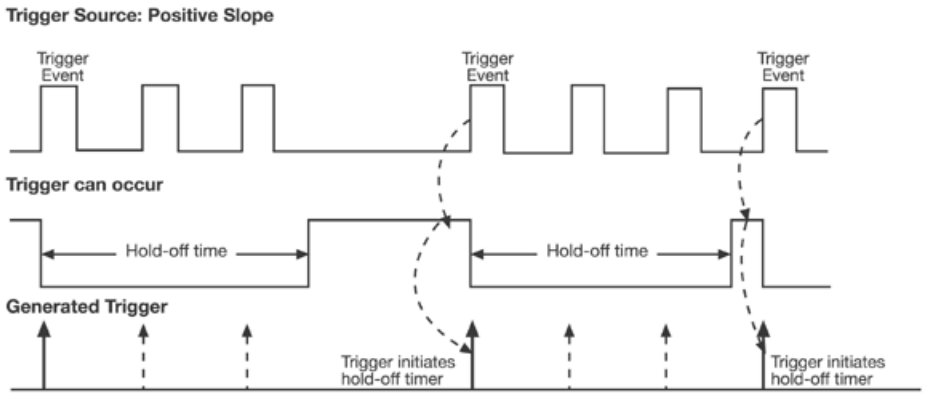
Use holdoff to obtain a stable trigger for repetitive, composite waveforms. For example, if the number or duration of sub-signals is known, you can disable them by choosing an appropriate holdoff value. Qualified triggers operate using conditions similar to holdoff.

Hold Off by Time

This is a period of time to wait to fire the trigger, either since the beginning of the acquisition or since the trigger conditions were met.

Sometimes you can achieve a stable display of complex, repetitive waveforms by placing a holdoff condition on the time between each successive Edge trigger event. This time would otherwise be limited only by the input signal, the coupling, and the instrument's bandwidth. Select a positive or negative slope, and a minimum time between triggers.

In the figure below, the bold edges on the trigger source indicate that a positive slope has been selected. The broken upward-pointing arrows indicate potential triggers, which would occur if other conditions are met. The bold arrows indicate where the triggers actually occur when the holdoff time has been exceeded.

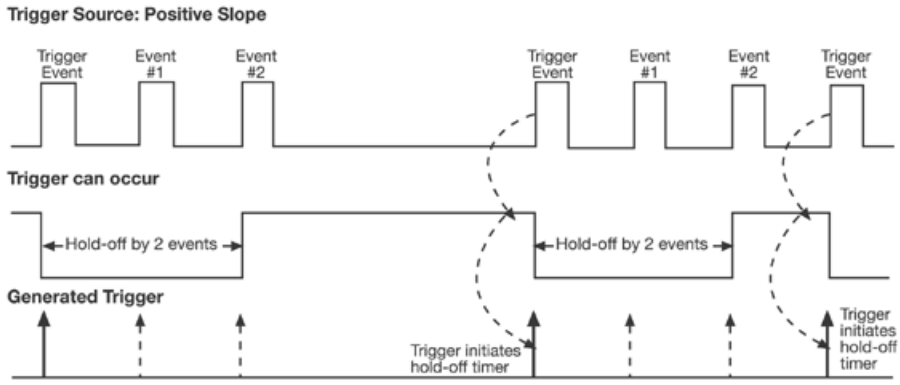


Edge trigger with holdoff by time.

Hold Off by Events

For purposes of Hold Off, Events refers to the number of times the trigger conditions have been met, counted either from the beginning of the acquisition or since the last trigger. For example, if the hold-off number of Events is 2 counted from the beginning of the acquisition, the trigger fires on the third event.

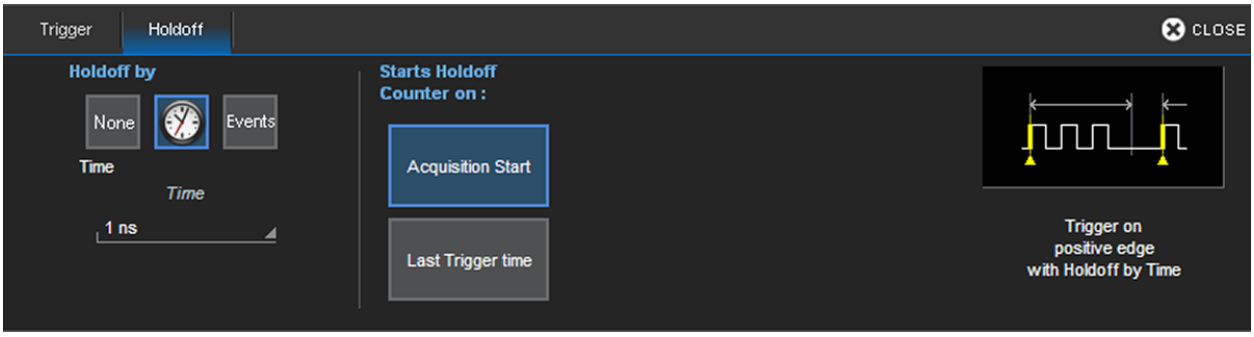
In the figure below, the bold edges on the trigger source indicate that a positive slope has been selected. The broken, upward-pointing arrows indicate potential triggers, while the bold ones show where triggers actually occur after the holdoff expires.



Edge trigger with holdoff by events.

Holdoff Settings

To access the Trigger Holdoff dialog, choose **Triggers > Trigger Setup** from the menu bar or press the front panel Trigger Setup button, then touch the **Holdoff tab**.



Choose to **Holdoff by** Time (clock) or Event. None disables Holdoff.

- 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 count before triggering.

Choose to **Start Holdoff Counter On** either:

- **Acquisition Start**, best for single-shot acquisitions.
- **Last Trigger Time**, best for acquiring repetitive waveforms.

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.

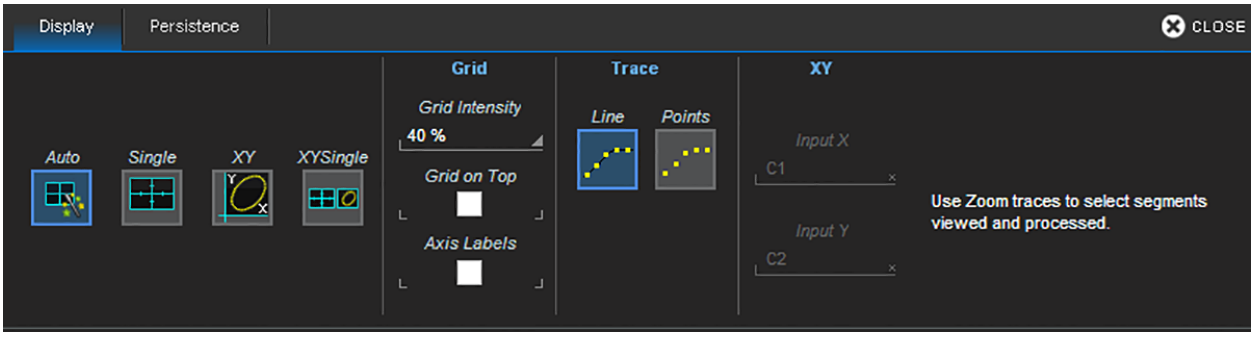
By default, the oscilloscope has **Auto Grid** enabled. This divides the screen into a maximum of three grids, one each for channels/memories, math functions, and zooms. All traces of the same type appear on the same grid.

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

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.

Display Settings

To access the Display dialogs, choose **Display > Display Setup** or **Display > Persistence Setup**.



Grid

Select one of the grid types:

Auto, the default, automatically adds or deletes grids as you open or close traces, up to the maximum number supported.

Single displays a single grid shared by all traces.

XY displays an XY type trace instead of a traditional voltage/time trace.

XY Single displays a single grid with an XY trace next to it.

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.

Check **Axis labels** to display the voltage 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

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

XY

XY displays plot 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.

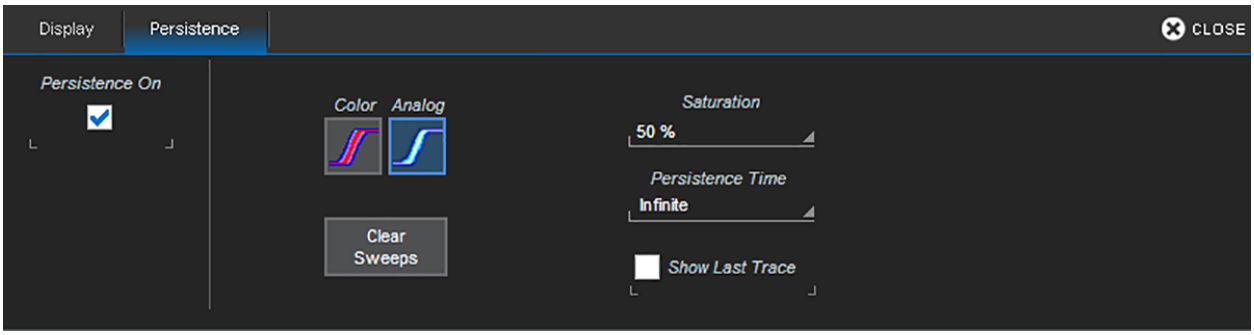
NOTE: The inputs can be any combination of channels, math functions, or memories, but both sources must have the same X-axis scale.

If you choose to display an XY grid, select the source channels to **Input X** and **Input Y**.

Persistence

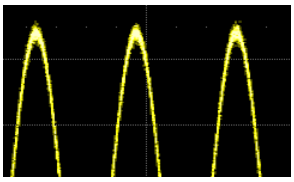
The Persistence feature retains waveform traces on the display for a set amount of time before allowing them to gradually "decay," similar to the display of old phosphor screen oscilloscopes. Use Persistence to accumulate on-screen points from many acquisitions to see your signal change over time. The persistence modes show the most frequent signal path in three-dimensional intensities of the same color (Analog), or graded in a spectrum of colors (Color). You can show persistence for any channel, math function, or memory.

Access the Persistence dialog by choosing **Display > Persistence Setup**. Check **Persistence On** to shown persistence, then select the mode, saturation level, persistence time, and last trace display.



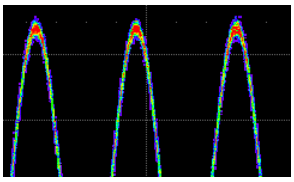
Persistence Mode

The Persistence display is generated by repeated sampling of the amplitudes of events over time, and the accumulation of the sampled data into display maps. These maps create an analog-style display. Statistical integrity is preserved because the duration (decay) is proportional to the persistence population for each amplitude or time combination in the data.



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.



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.

Other Persistence Settings

Besides the different modes, you can select a **Saturation** level as a percentage of the maximum population. All populations above the saturation population are then 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.

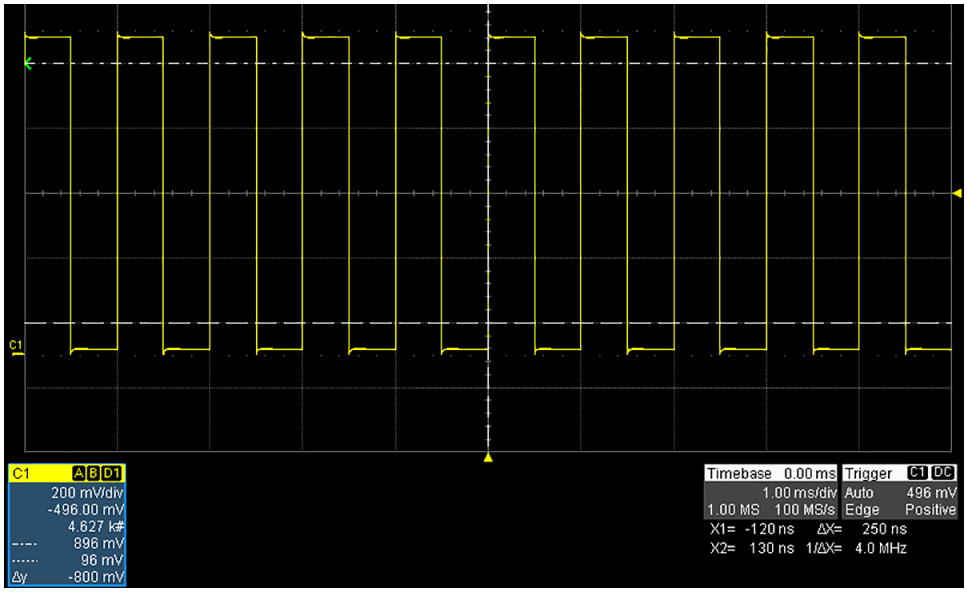
Persistence Time is the duration of time (in seconds) after which persistence data is erased from the display.

Choose to superimpose the last waveform over the persistence display by selecting **Show Last Trace** .

Cursors

Cursors are markers (lines or cross-hairs) that identify specific 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 descriptor box for the trace; Horizontal (time) cursor readouts appear below the Timebase descriptor box.



Horizontal and vertical cursors.

Cursor Types

Standard Cursors

These cursors can be placed on most any Channel, Memory, Math or Zoom trace.

Horizontal (Time) cursors place vertical lines through a desired point along the horizontal axis.

Vertical (Amplitude) cursors place horizontal lines through a point on the vertical axis.

An option exists to place **Horizontal + Vertical** cursors together.

Special Cursors

Some 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.

In addition, some optional software packages provide cursors and help markers that are specific to the application.

Cursor Settings

Display Cursors

To quickly turn on/off cursors, either:

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

Position Cursors

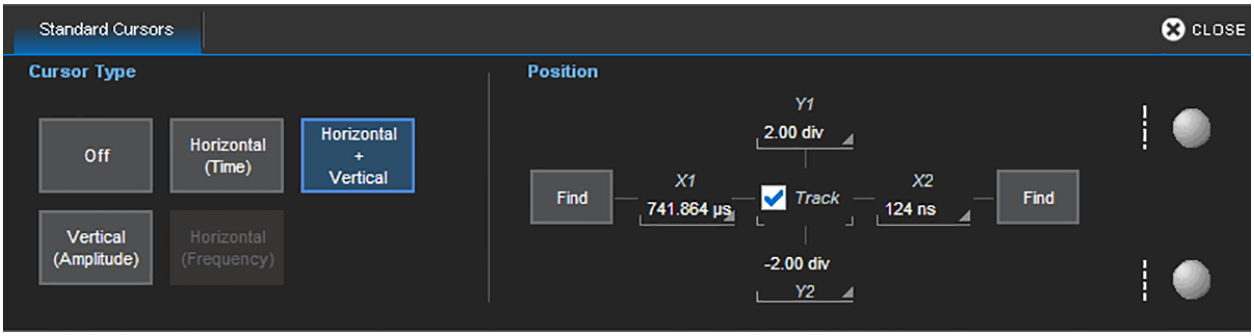
Te easiest way to position a cursor is to touch and drag the cursor line to a new position.

Alternatively, with the cursor on, turn the front panel **Cursors** knob. If there is more than one cursor line, push the Cursor knob until the correct line is selected, then turn the knob to move it.

Use the **Position** data entry controls on the Standard Cursors dialog to place cursors precisely.

Standard Cursors Dialog

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



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

The **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.

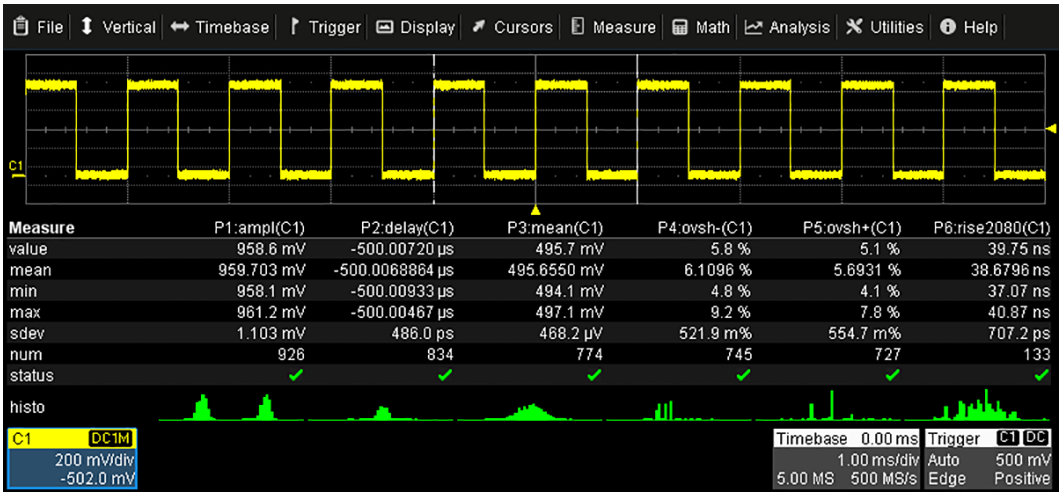
- **X 1** (negative) and **X 2** (positive) time from the zero point.
- **Y 1** (negative) and **Y 2** (positive) number of divisions from the zero level. May be a fraction of a division.
- **Track** locks cursor lines so they move together, maintaining their same relative distance from each other.

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.

You can create a custom set of up to six parameters drawn from all the standard measurements, as well as specialized measurements installed with optional software packages.

Measurement readouts appear in a table below the grid. Readouts can be individually turned on/off. To quickly access the Measure Setup dialog if it is closed, touch any cell of the readout table.

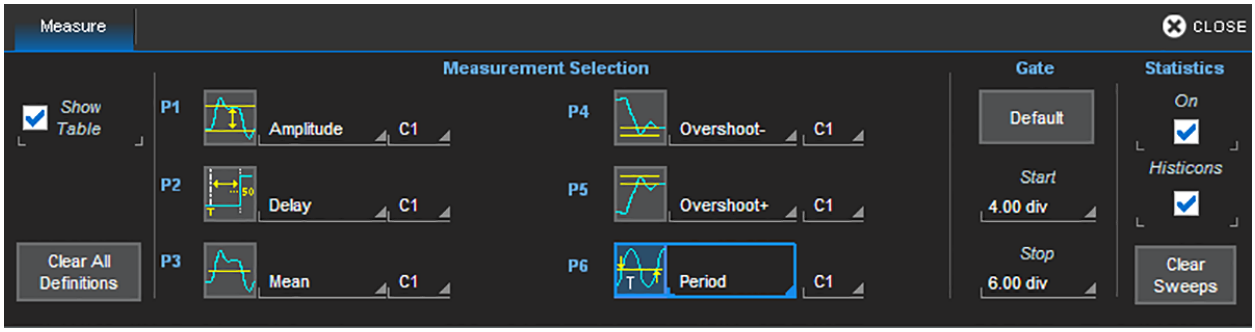


Measurement readout table open below grid showing statistics and histicons.
(Histicons available with the WS10-ADT option.)

Set Up Measurements

To configure custom measurements to add to the table of parameter readouts:

1. From the menu bar, choose **Measure > Measure Setup**.
2. Check **Show Table** to display the readout on screen.
3. Touch the **Measure button** next to an unused **Px** location (or one that you want to change), then choose the measurement from the pop-up window.



4. For each input required by the measurement, touch **Source** and select the waveform to be measured.
5. Optionally, set a measurement gate by entering the Gate **Start** and **Stop** divisions or dragging the gate posts from the far left and right edges of the grid to reposition them.
6. Choose to display **Statistics** or **Histicons** (WS10-ADT only) on the measurement readout table.

Gating Measurements

By using gates, you can narrow the span of the waveform on which to perform parameter measurements, allowing you to focus on the area of greatest interest. For example, if you "gate" five rising edges of the waveform, the parameter calculations for rise time 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. The gate, therefore, initially encloses the entire waveform.

The quickest way to set a gate is to drag the gate posts located at the far left and right of the grid to the desired positions. You can refine this setting by specifying a position down to hundredths of a division in the **Gate Start** and **Stop** fields on the Measure dialog dialog. All parameters share the same gate.

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

Viewing Statistics

You can add the statistical measures **value(last)**, **mean**, **min.**, **max.**, **sdev**, and **num**(ber of measurements computed) to the measurement parameter readout table by checking **Statistics On**. You can also choose **Measure > Statistics** from the menu bar.

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

For any parameter that computes on every event, the value displayed is equal to the number of events per acquired waveform. If *x* waveforms were acquired, the value represents *x* times the number of cycles per waveform. The value(last) statistic is equal to the measurement of the last cycle on the last acquisition.

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

Viewing Histicons

The histicon capability is available on WaveSurfer 10 oscilloscopes with the WS10-ADT option installed.

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.

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.

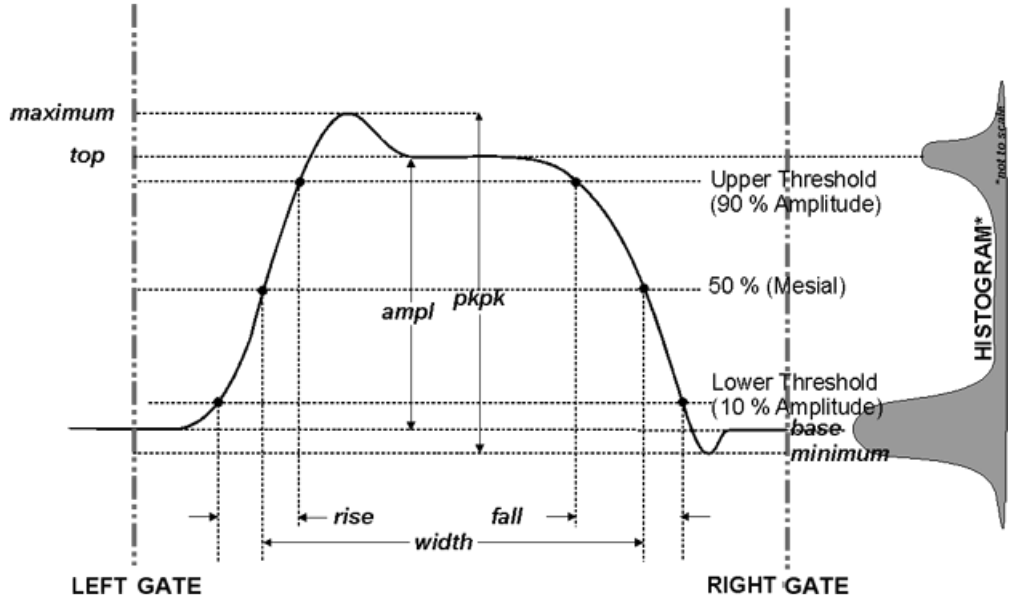
Measurement	Description
Amplitude (ampl)	Measures the difference between upper and lower levels in two-level signals. Differs from pkpk in that noise, overshoot, undershoot, and ringing do not affect the measurement. Amplitude is calculated by using the formula Top – Base. On signals not having two major 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 (base - min.)/ampl x 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 (max. - top)/ampl x 100. On signals not having two major levels (triangle or saw-tooth waves, for example), may not give predictable results.
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 *maximum – minimum*.

Measurement	Description
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: $\sqrt{\frac{1}{N}\sum_{i=1}^{N}(v_i - mean)^2}$ 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

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, calculation of the rise and fall times is easily done (see figure). The appropriate threshold levels are automatically determined by the instrument, using the amplitude (ampl) parameter.

Rising Edge Duration

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

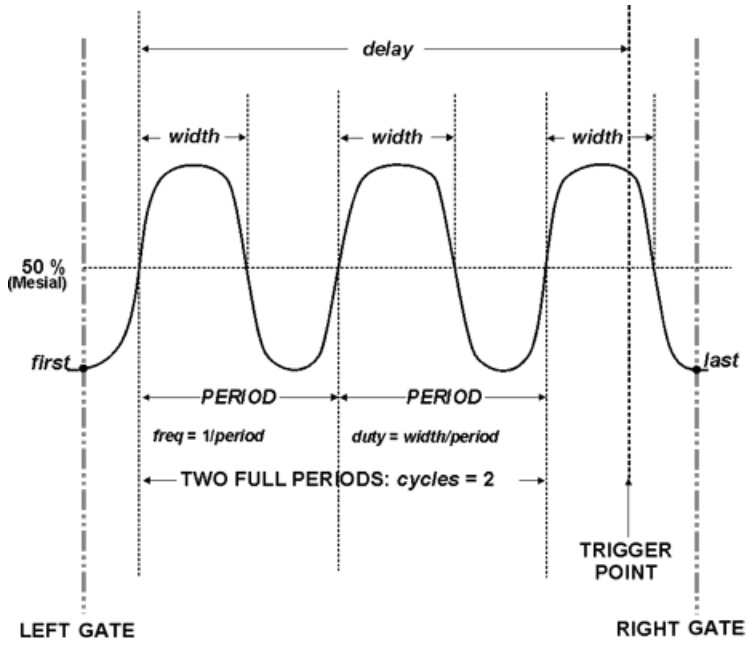
Falling Edge Duration

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

Where *Mr* is the number of leading edges found, *Mf* the number of trailing edges found, Tr_i^x the time when rising edge *i* crosses the x% level, Tf_i^x and 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.



Math

Math traces) 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.

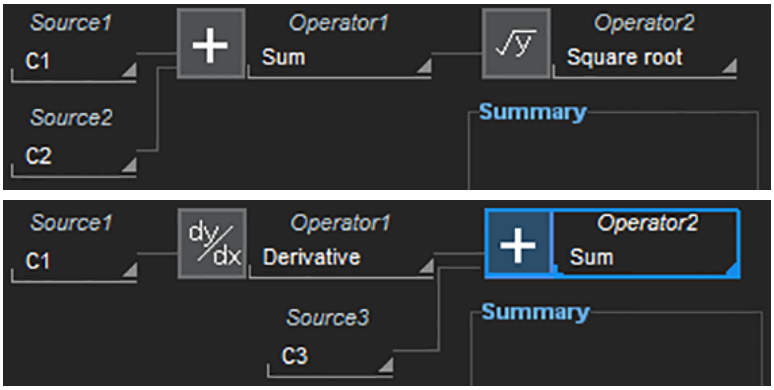
Math can be applied to any channel (C*x*), zoom (Z*x*), or memory (M*x*) trace. On oscilloscopes with the WS10-ADT option, it can even be applied to another math trace (F*x*), allowing you to chain operations (for example, trace F1 can show the average of C1, while trace F2 provides the integral of F1). Functions such as Trend can be applied to measurement parameters (P*x*) to plot the history of the measurement.

In addition to the extensive math capabilities that are standard with every oscilloscope, 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 datasheets on the Teledyne LeCroy website at teledynelecroy.com. If you have installed software options, these capabilities are accessed through the oscilloscope Analysis menu, rather than the Math menu, although special measure parameters and math functions will be available when using Measure and Math dialogs.

Single vs. Dual Operator Functions

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).



Set Up Math Function

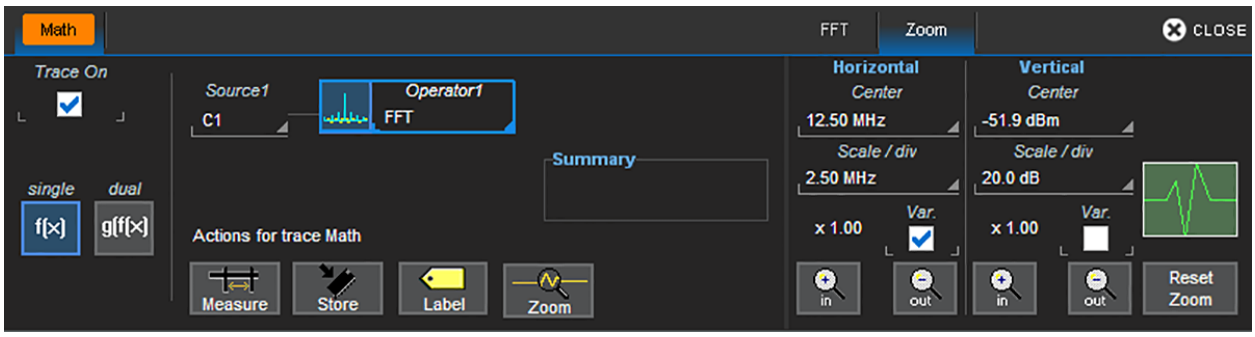
This procedure explains how to set up math function (F*x*) traces. Function traces take as input one or more channel, zoom, memory or math traces and output a new math trace.

NOTE: You can set up two, single- or dual-operator math functions on WaveSurfer 10 oscilloscopes with the WS10-ADT option installed. Use the F1 and F2 dialogs to configure the functions; use the Math dialog to enable/disable them.

1. From the menu bar, choose **Math > Math Setup**.

   TIP: If you know which function number you'll be using, select **F*x* Setup** right from the Math menu.

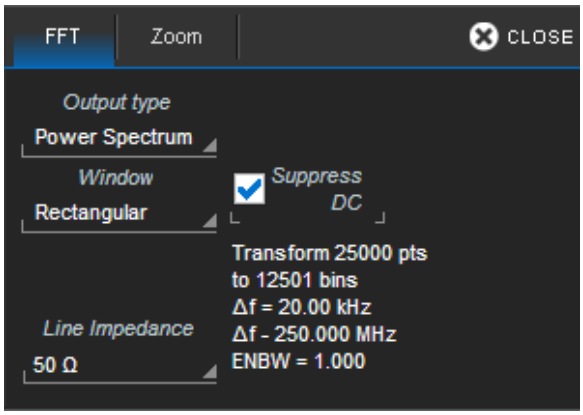
2. If you have the WS10-ADT option installed, choose a location by touching one of the **F*x* tabs**.



3. Choose a **single f(x)** or **dual g(f(x)** operator function.
4. In **Operator1**, choose the math operation to perform.
5. The choice of operator drives the number of **Source** fields you will see displayed. Make a selection in each 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).

6. If the operator you've selected has any other configurable settings, you'll see a right-hand dialog 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.



There will also be a Zoom dialog where you can adjust the math trace Vertical range. This does not affect the scale of any other traces.

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. See the specifications at teledynelecroy.com.
Derivative	Calculates the derivative of adjacent samples using the formula: *(next sample value – current sample value) / (horizontal sample interval)*
DIfference	For every point in the waveform, 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 1 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: *(current sample value + next sample value) * (horizontal sample interval)* Each calculated area is summed with the previous sum of areas. The multiplier and adder are applied before the integration function.
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 / (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.

Operator	Definition
Roof	Calculates the highest vertical values of a waveform at each horizontal value for a specified number of sweeps.
Square	For every point in the waveform, the square of the sample value is calculated.
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.
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.

Advanced Debut Toolkit Math Functions

These operators are added with the WS10-ADT option.

Operator	Definition
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. See the specifications at teledynelecroy.com.
ERes	Applies a noise reduction and smoothing filter by adding a specified number of bits. See Enhanced Resolution.
Exp	Calculates the antilog to the base e of the source; that is, e raised to the power equal to the source.
Exp10	Same as Exp, using base 10.
Invert	For every point in the waveform, the inverse of that point is calculated.
Ln	Peforms a natural log of a waveform. Values less than or equal to zero are set to underflow.
Log10	Performs a log base 10 of a waveform. Values less than or equal to zero are set to underflow.
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.
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. See Trend.

FFT

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.

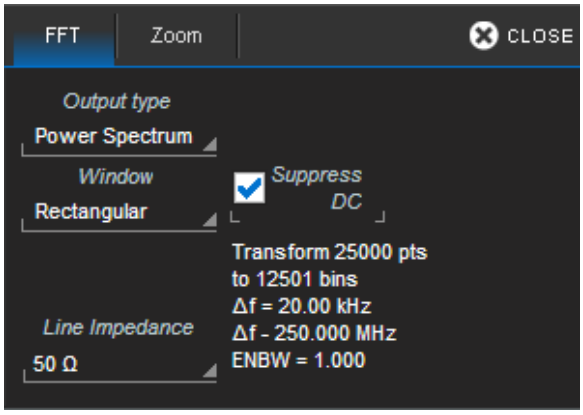
If sampling is done at a rate fast enough to faithfully approximate the original waveform (usually five times the highest frequency component in the signal), the resulting discrete data series will uniquely describe the analog signal. This is of particular value when dealing with transient signals because, unlike FFT, conventional swept spectrum analyzers cannot handle them.

Because of its versatility, FFT analysis has become a popular analysis tool. However, 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 of the oscilloscope and therefore determines the frequency resolution and span at which spectral analysis can be carried out.

Set Up FFT

1. Follow the usual steps to set up a math function, selecting **FFT** from the **Frequency Analysis** submenu.
2. Open the **FFT** right-hand dialog.



3. Choose an **Output type**.
4. Optionally, choose a weighting **Window**. See below for more information about FFT weighting windows.
5. Depending on your **Output Type** selection, you may also make selections for **Line Impedence**. By default, the FFT function assumes that the oscilloscope is terminated in 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.
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 (sinx)/x function in the spectral domain, slight attenuation will be induced. Functions with less attenuation (Flat Top and Blackman-Harris) provide maximum amplitude at the expense of frequency resolution, whereas Hamming and Von Hann are good for general purpose use with continuous waveforms.

Window Type	Applications and Limitations
Rectangular	These are 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)	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Hamming	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Flat Top	This window provides excellent amplitude accuracy with moderate reduction of leakage, but with reduced frequency resolution.
Blackman-Harris	It reduces the 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

Averaging Waveforms

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 NORM/AUTO to STOP. The instrument resumes averaging when you change the trigger mode back to NORM/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:

new average = (new data + weight * old average)/(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**:

1st sweep (no old average yet): new average = (new data +0 * old average)/(0 + 1) = new data only

2nd sweep: new average = (new data + 1*old average)/(1 + 1) = 1/2 new data +1/2 old average

3rd sweep: new average = (new data + 2 * old average)/(2 + 1) = 1/3 new data + 2/3 old average

4th sweep: new average = (new data + 3 * old average)/(3 + 1) = 1/4 new data + 3/4 old average

5th sweep: new average = (new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

6^{th} sweep: new average = (new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

7^{th} sweep: new average = (new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 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.

Set Up Averaging

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

To apply Continuous or Summed Averaging as a Math function:

1. Follow the usual steps to set up a math fuction, selecting **Average** from the **Basic Math** submenu.
2. On the **Average** right-hand dialog, choose **Summed** or **Continuous**.
3. Touch **Sweeps** and provide a value. The valid range is 1 to 1,000,000 sweeps.

Enhanced Resolution

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).

ERes can be applied as a form of Pre-Processing, or as a Math function.

Set Up Enhanced Resolution (ERes)

To quickly set up ERes, open the Channel setup dialog and in the Pre-Processing section select a **Noise Filter (ERes)** bit size .

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. Touch the **ERes** right-hand dialog tab , then touch **bits** and make a selection 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 your signal is single-shot or repetitive. The signal-to-noise ratio (SNR) improvement you gain is dependent 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. The parameters of the six filters are given in the following table.

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.

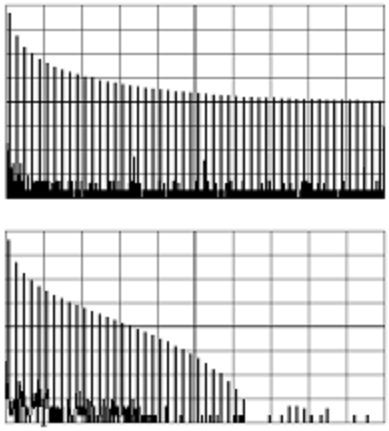
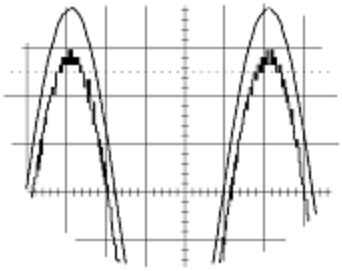
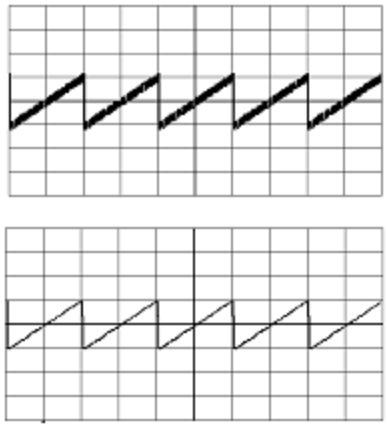
The opposite may be true if the noise is mostly at lower frequencies. SNR improvement due to the removal of coherent noise signals - feed-through of clock signals, for example - is determined by the fall of the dominant frequency components of the signal in the passband. This is easily ascertained using 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. Enhanced resolution 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.

Example ERes Applications

The following examples illustrate how you might use the instrument's enhanced resolution function.

Graph	Function
	**In low-pass filtering:** The spectrum of a square signal before (left top) and after (left bottom) enhanced resolution processing. The result clearly illustrates how the filter rejects high-frequency components from the signal. The higher the bit enhancement, the lower the resulting bandwidth.
	**To increase vertical resolution:** In the example at left, the lower (inner) trace has been significantly enhanced by a three-bit enhanced resolution function.
	**To reduce noise:** The example at left shows enhanced resolution of a noisy signal. The original trace (left top) has been processed by a 2-bit enhanced resolution filter. The result (left bottom) shows a smooth trace, where most of the noise has been eliminated.

NOTE: While enhanced resolution can only improve 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

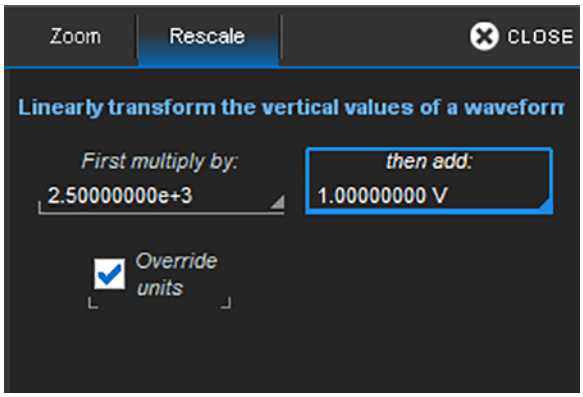
lengths. Data is lost at the start and end of the waveform and the trace ends up slightly shorter after filtering. The number of samples lost is exactly equal to the length of the impulse response of the filter used: between 2 and 117 samples. Normally this loss (just 0.2 % of a 50,000 point trace) is 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.

Rescaling and Assigning Units

The rescale function allows you to apply a multiplication factor (*a*) and additive constant (*b*) to your waveform: *a*X + *b*. You can do it in the unit of your choice, depending on the type of application.

Set Up Rescaling

1. Follow the usual steps to set up a math function, selecting **Rescale** from the **Functions** submenu.
2. Touch the **Rescale** right-hand dialog tab.



3. To apply a multiplication factor:
   - Check the **First multiply by:** box and enter a value for *a*, the multiplication factor.
   - Touch **then add:** and enter a value for *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 abbreviation for the unit the measure you wish to use.

   You can also enter combinations of the unit abbreviations 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: S2 = seconds squared.

   NOTE: Some units may be converted to simple units (e.g., V.A will display as W).

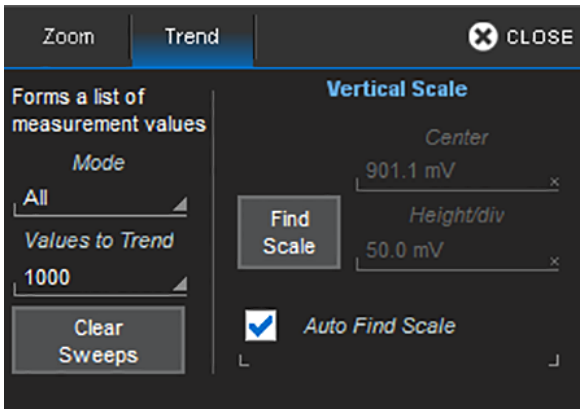
Abbreviated Units of Measure

Abbreviation	Measure	Abbreviation	Measure
(blank)	No units	**N**	Newton
A	Ampere	**OHM**	Ohm
C	Coulomb	**PAL**	Pascal
CYCLE	Cycles	**PCT**	Percent
DB	Decibel	**POISE**	Poise
DBC	Decibel referred to carrier	**PPM**	Parts per million
DBM	Decibel Milliwatt	**RAD**	Radian
DBV	Decibel Volts	**DEG**	Degree (of arc)
DBUZ	Decibel Microamp	**MNT**	Minute (of arc)
DEC	Decade	**SAMPLE**	Sample
DIV	Divisions	**SWEEP**	Sweeps
Event	Events	**SEC**	Second (of arc)
F	Farad	**S**	Second
G	Gram	**SIE**	Siemens
H	Henry	**T**	Tesla
HZ	Hertz	**UI**	Unit interval
J	Joule	**V**	Volt
K	Degree Kelvin	**VA**	Volt amps
CEL	Degree Celsius	**W**	Watt
FAR	Degree Fahrenheit	**WB**	Weber
L	Liter	**MIN**	Min
M	Meter	**HOUR**	Hour
FT	Foot	**DAY**	Day
IN	Inch	**WEEK**	Week
YARD	Yard		
MILE	Mile		

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. Trends are especially useful for visualizing the history of a parameter over an extended period of time or over multiple acquisitions.

1. Follow the usual steps to set up a math function, selecting **Trend** from the submenu.
2. Open the **Trend right-hand dialog**.



3. Choose a computation **Mode** of All (plots multiple points per acquisition) or Average (plots one point per acquisition).
4. To rescale the Trend plot, uncheck **Auto Find Scale** and enter the new **Center** and **Height/div** values. You can also use **Find Scale** to automatically find suitable values.

Memory

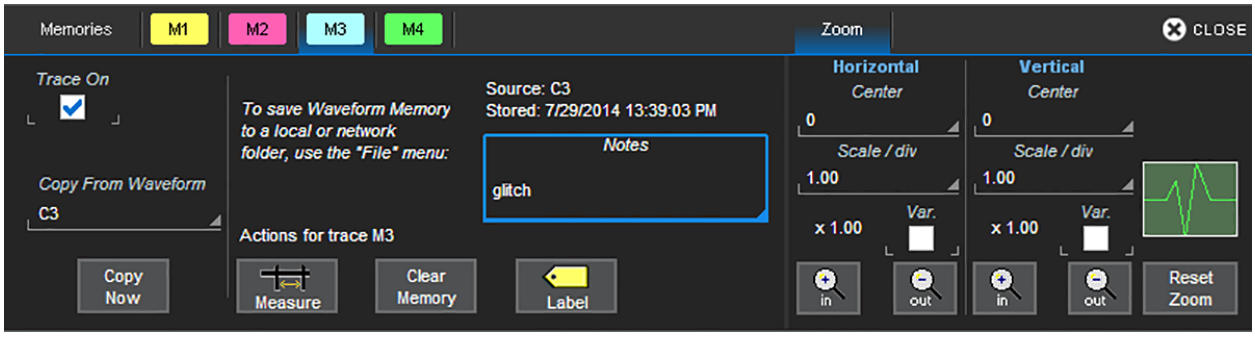
The oscilloscope is equipped with fourinternal memory slots (M*x*) to which you can copy any channel, math, zoom or other special waveform that is active on the grid. This is a convenient way to store an acquisition for later viewing and analysis.

Memories are created at the same scale as the source trace, but they can be adjusted independent of the original by using the Zoom controls that appear next to the M*x* dialogs.

Save Waveform to Memory

1. With the source waveform displayed on the grid, press the front panel **Mem button** or choose **Math > Memory Setup** to open the Memories dialog.
2. Touch the **M*x* tab** corresponding to the memory slot you wish to use.

   NOTE: 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.



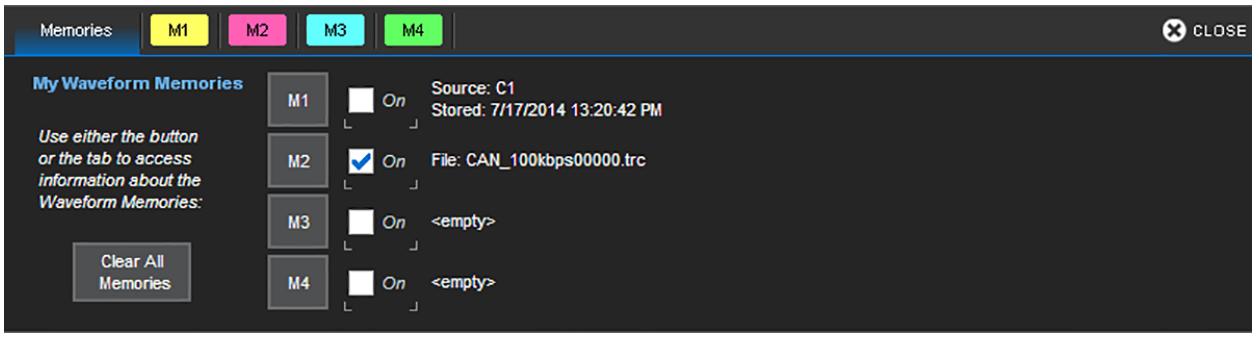
3. In **Copy from Waveform**, choose the source trace to copy to memory.
4. Touch **Copy Now**.
5. Optionally, check **Trace On** to immediately display the memory. Use the Zoom controls to adjust the scale of the memory while it is turned on.

Save Waveform Files to Memory

Trace (.trc) files saved on other Teledyne LeCroy oscilloscopes can also be stored to internal memory. Use the Recall Waveform function to save external files to memory. Then, you can use the Memories dialog to enable them from the oscilloscope.

Restore Memory

1. Access the Memories dialog by pressing the front panel **Mem button** or choosing **Math > Memory Setup**.
2. Check **On** next to the memory you wish to display. A description of the memory showing the source channel and creation time appears next to each M*x* on the dialog.



Analysis

Most Teledyne LeCroy oscilloscopes calculate measurements for all instances in the acquisition, enabling you to rapidly and thoroughly analyze a long memory acquisition of thousands or millions of parameter values, or to apply a variety of mathematical functions to the waveform trace.

WaveScan searches a single acquisition for events that meet specific criteria, enabling you to zoom in on anomalies in the waveform, or scans multiple acquisitions with allowable trigger actions when conditions are met. It can also be used to filter measurements. A variety of views help you understand the behavior of waveforms.

Pass/Fail Testing indicates whether or not waveforms meet a set of defined criteria.

Optional software packages may be purchased that simplify specialized analysis, such as various Serial Data Decode options. These add new functionality to the oscilloscope Analysis menu.

WaveScan

The WaveScan® Search and Find tool enables you to search for unusual events in a single capture, or to scan for a particular event in many acquisitions over a long period of time. Each Scan Mode is optimized to find a different type of event. Results are time stamped, tabulated, and can be selected individually.

WaveScan window with different scan "views" turned on.

Customize the presentation by choosing different WaveScan display features, or Scan Views. Optionally, set Actions to occur automatically when unusual events are found, such as stopping the acquisition or sounding an alarm.

NOTE: Whenever WaveScan is enabled, the instrument reverts to Real-time sampling mode.

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. Additional settings for Edge Mode 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. A marker displayed over the source trace indicates the 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. Additional settings for Non-monotonic Mode are:

- **Slope**. Choose Pos, Neg, or Both.
- **Hysteresis is** (set in...). Choose Division, Percent, Absolute.
- **Division/Percent/Absolute**. Enter the hysteresis level in the units you selected.
- **Levels are** (set in...). Choose Percent, Absolute, or Pk-Pk%.
- **High Level** and **Low Level**. Enter the top and bottom thresholds in the units you selected.

Runt Mode

Runt Mode looks for pulses that fail to cross a specified threshhold. You can search for positive-going or negative-going runts, or both. An adjustable hysteresis band is provided to eliminate noise.

In the case of negative-going runt pulses, the value displayed in the table is the difference (delta) of the high level of the signal and the runt amplitude (i.e., where the runt bottoms out). This can be confirmed by placing cursors on the runt pulse and reading the delta Y value in the trace labels. In the case of positive-going runt pulses, the value displayed in the table is the absolute value of the amplitude of the runt pulse. Additional settings for Runt Mode are:

- **Runt Type**. Choose Both, Pos, or Neg.
- **Hysteresis**. Enter the hysteresis level as a percentage or voltage.
- **Low Threshold** and **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. Additional Settings for Measurement Mode 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) is used for finding 2- to 16-bit patterns across the digital lines. Additional settings for Bus Pattern Mode are:

- **Viewing**. Choose to enter the pattern as Binary or Hex(adecimal).
- **Binary/Hex**. Enter the pattern.
- **Num. Patterns to detect**. Enter a whole number.

Scan Views

Scan Views are different ways to view your WaveScan results. You can choose to display views simultaneously or visit them sequentially. Just check the boxes at the bottom of the WaveScan dialog for those views you wish to display. Uncheck the box to turn off the view.

By default, the **Source Trace** is displayed in the top grid, with markers indicating points in the trace that meet the search criteria.

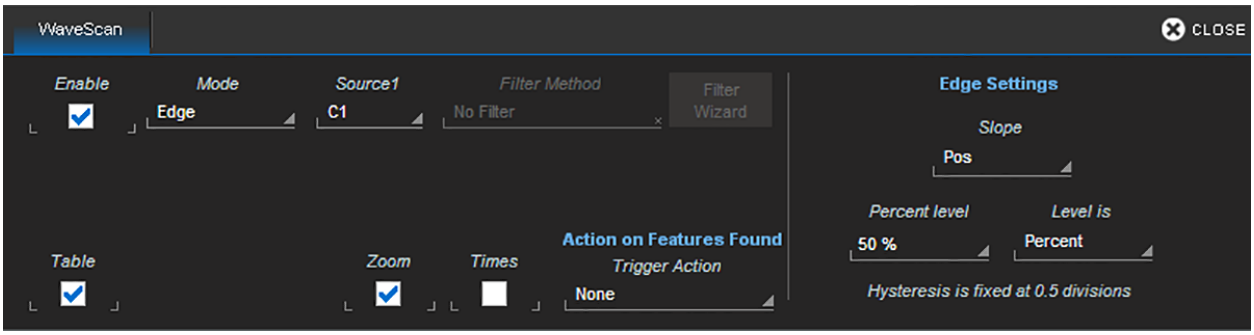
Table view displays a table of measurements relevant to your chosen Search Mode next to the source trace. **Times** view adds columns to the table showing Start and Stop Times for each event.

Zoom view works exactly as it does elsewhere in the oscilloscope software, opening a close-up of the source trace in a new grid that you can adjust vertically and horizontally. A Z*x* tab appears by default when you launch WaveScan; see Zoom Controls for an explanation of the remainder of the controls found on this dialog.

A unique feature of the WaveScan Zoom is that you can automatically zoom the events captured from the source trace by touching the Prev/Next buttons on the Z*x* dialog. You can also select the event from the Table display, and you are automatically relocated to that event on the zoom trace.

Setting Up WaveScan

This procedure explains how to set up WaveScan to search an acquisition for events of interest. 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. If you're using Measurement Mode, set up the filter in one of the following ways:
   - Touch **Filter** and choose an operator, then enter the **Filter Limit**.
   - Touch **Filter Wizard** and choose one of the pre-set filters. The Filter and Filter Limit are automatically set based on your selection.
7. 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.
8. Optionally, choose an **Action** to trigger when an event that meets your scan criteria is found.
9. Restart acquisition.
10. When using the Zoom view, use the Z*x* tab to adjust the zoom.

Pass/Fail Testing

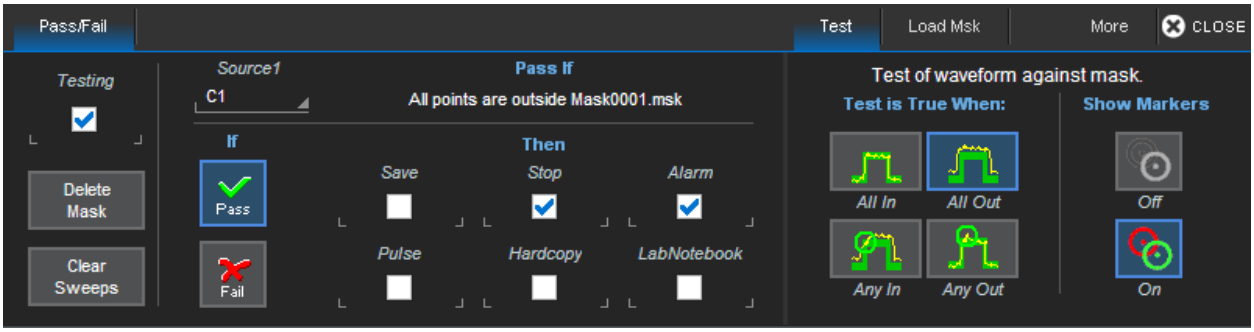
Pass/Fail testing is a type of mask testing that is particularly useful for comparing newly acquired signals to a previously acquired "golden standard" waveform.

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. The mask test can be confined to just a portion of the trace by the use of a measure gate.

Access Pass/Fail Test Dialogs

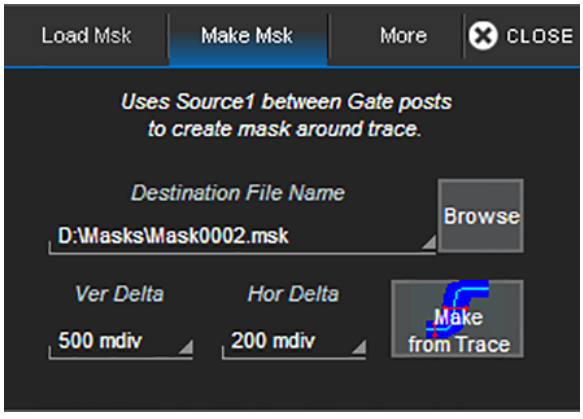
Choose **Analysis > Pass/Fail** to display the **Pass/Fail** dialog.



Make Mask

Use this procedure to create a new mask based on a live waveform. The mask will cover the area of the waveform, plus the boundary values you enter.

1. Touch the **Make Mask tab** to display the dialog.

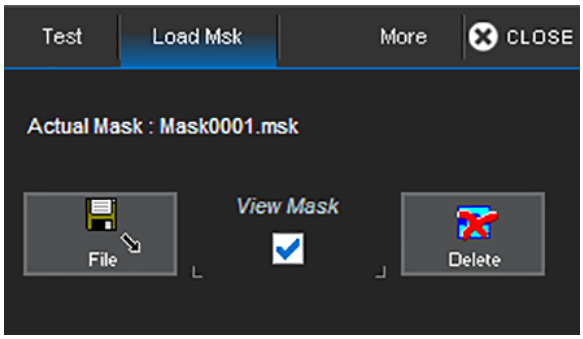


2. If desired, enter a new **Destination File Name** and path, or touch Browse and select a previous file to overwrite. The file name should end with the **.msk** extension.
3. Touch the **Ver Delta** and **Hor Delta** fields and enter boundary values using the pop-up numeric keypad or the front panel 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, or wish to recall a mask you previously created and saved.

1. Touch the **Load Mask tab** to display the dialog.

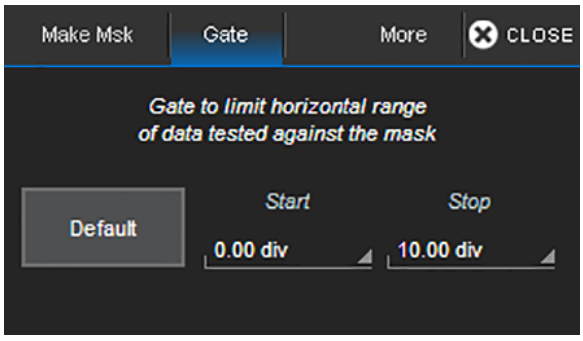


2. To use a saved .msk file, touch **File** and select the mask.
3. Check **View Mask** to display the mask over the trace.

Set Gates

Optionally, set gates to limit the portion of the waveform that is compared to the mask.

1. Touch **More** to display the Gate tab, then open the **Gate** dialog.

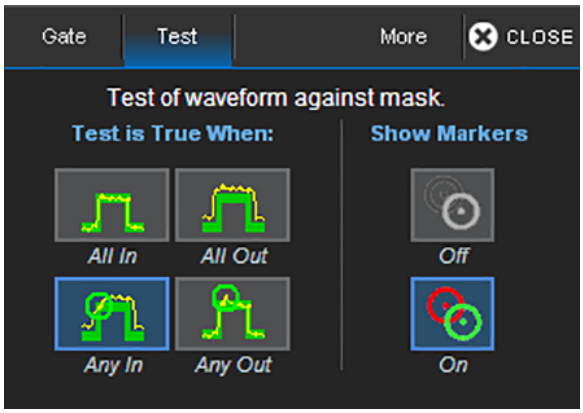


2. Enter the **Start** and **Stop** horizontal divisions that mark the segment of the waveform to be tested with this mask. This can be a whole division or a fraction of a division. Divisions are numbered 1-*n* left to right.

   Tip: A quick way to position the gate is to drag the gate posts initially placed at the extreme left and right ends of the grid to the desired points.

Define Test

1. Touch **More** to display the Test tab, then open the **Test** dialog.



2. Select one of the conditions that, when True (yes), result in a Pass.
3. Optionally, turn **Off/On** markers. Markers visually indicate where on the waveform mask violations have occurred.

Run Test

1. On the main Pass/Fail dialog, select the **Source** trace to test.
2. Select any additional actions for the oscilloscope to take in the event of a **Pass** or **Fail**.

   Save the waveform to a file.

   Stop the test.

   Sound an **Alarm**.

   Send a **Pulse** via the Aux Out connector.

   Capture the screen and process it according to your **Hardcopy** setting (print, email, or save it to file).
3. Select the **Testing** checkbox to start the test.

   A test counter showing the number of sweeps and the number that Passed or Failed appears below the grid area. If you have turned on markers, the source waveform samples that Failed the mask test are overlaid in red on the grid; the samples that Passed the test are overlaid in green.

Removing a Mask from the Display

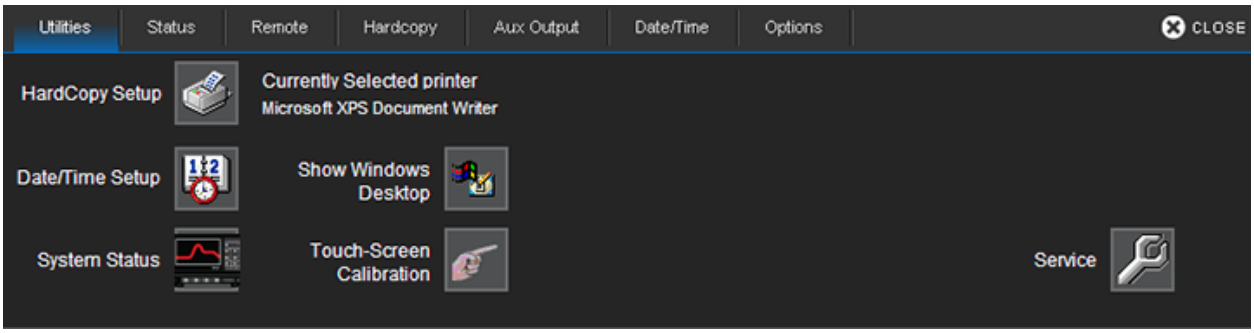
1. Access the Pass Fail dialogs.
2. Touch the **Delete Mask** button.

Utilities

Utilities

Utilities settings primarily control the instrument's interaction with other devices/systems. Preferences, on the other hand, tend to control the appearance and performance of the oscilloscope application.

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



HardCopy Setup, Date/Time Setup, and System Status buttons open their corresponding dialogs, as do the tabs.

There are also tabs linking to Remote Control, Auxilliary Output, and Options settings.

Touch-Screen Calibration launches a sequence of display calibration screens. You will be prompted through a series of actions to improve the precision and accuracy of the touch screen.

The **Service** button to the far right of the dialog launches a section of the application reserved for qualified Teledyne LeCroy service personnel. An access code is required to enter this section.

System 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

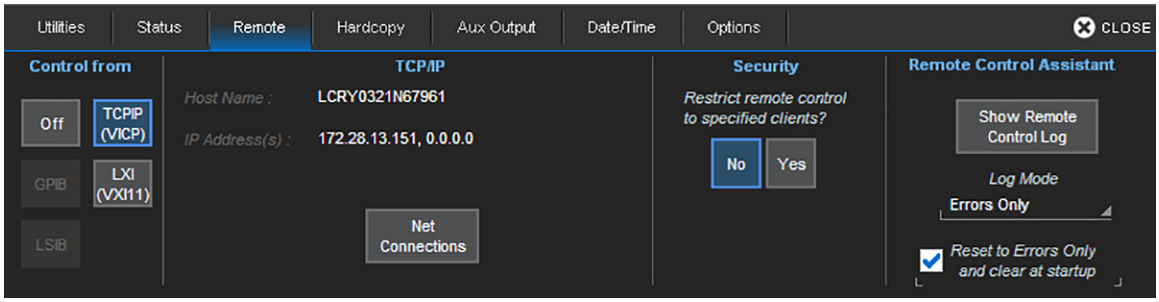
Choose **Support > About** from the menu bar.

The Utilities Status is not the same as the Status feature accessed through various menus (e.g., Vertical > Channels Status). That feature displays the current state of the oscilloscope configurations —such as acquisition, channel, measurement parameter, math function, and memory settings.

Remote Control Settings

The Remote dialog contains settings to configure remote control of the instrument. Supported 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 teledynelecroy.com. The instrument uses Dynamic Host Configuration Protocol (DHCP) as its default addressing protocol. You can assign a static IP address using the standard Windows network setup menus.
- **LXI (Ethernet)**
- **GPIB**. To activate this option, connect the GPIB-USB adapter to any host USB port.



Assign Static IP Address/Name Server

Before starting, consult with your Network Administrator regaring 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 **SMSC** icon.
5. On the Ethernet Driver Settings dialog, choose **Specify an IP Address**.

   If assigning a Name server, leave Obtain an IP address via DHCP.
6. Touch the **IP Address** field, and use the keyboard to enter the address. Repeat for Subnet Mask and Default Gateway.

   Alternatively, touch the Name Server tab and enter the DNS server address.
7. Touch the window close boxes to return to the oscillosocpe application.

Set Up Remote Control

1. From the menu bar, choose **Utilities > Utilities Setup**, then touch the **Remote tab**.
2. On the **Remote** dialog, make a **Control From** selection.
3. If using TCPIP or LXI, touch the **Net Connections** button and select a network from the pop-up.
4. If using TCPIP and wish to restrict control of the oscilloscope to specific network clients, touch **Yes**. Enter the IP addresses or DNS names of the authorized controllers in a comma-delimited list.

Configure the Remote Control Assistant Event Log

The **Remote Control Assistant** monitors communication between the controller and oscilloscope when you are operating the instrument remotely. You can log all events or errors only. The log can be output to an ASCII file and is invaluable when you are creating and debugging remote control programs.

1. From the menu bar, choose **Utilities > Utilities Setup**, then touch the **Remote tab**.
2. Under **Remote Control Assistant**, touch **Log Mode** and choose **Off**, **Errors Only**, or **Full Dialog**.
3. To always clear the log at startup, check **Reset to Errors Only and clear at startup**.

Export Contents of the Event Log

1. From the menu bar, choose **Utilities > Utilities Setup**, then touch the **Remote tab**.
2. Touch the **Show Remote Control Log** button. The Event Logs pop-up is shown.
3. Enter a log file name in **DestFilename**, or touch Browse and navigate to an existing file.

   NOTE: New contents will overwrite the existing content; it is not appended.
4. Touch **Export to Text File**.

Hardcopy (Print) Settings

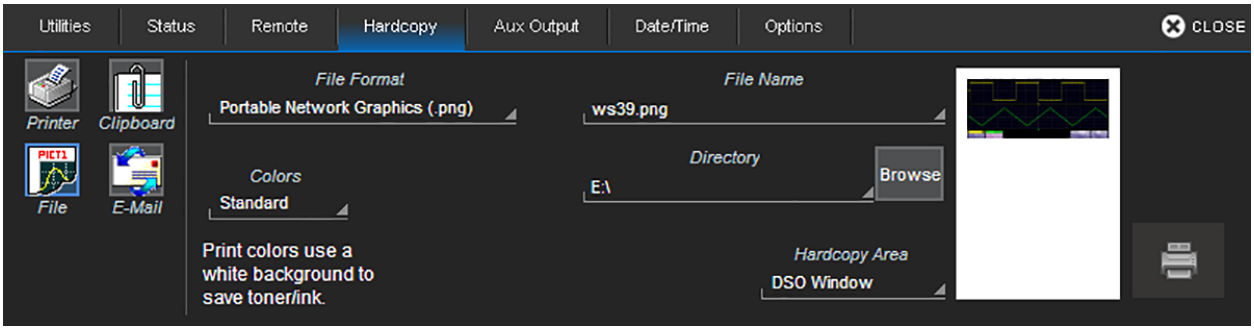
Hardcopy settings control how the oscilloscope **Print** function behaves. Print captures an image of the oscilloscope display, but there are several options as to what it does with the image next:

- Send to a hardcopy printer
- "Print" to a file that can be saved to an internal or external drive
- Send to E-Mail
- Copy to the Windows clipboard for you to paste elsewhere

Each option is set up on the Utilities Hardcopy dialog. You can further set up a default print color scheme and capture area. A preview of your hardcopy setup appears to the right of the dialog.

NOTE: You can configure the front panel Print button to create a new Notebook Entry to be included in a LabNotebook report. This is not done in Utilities Hardcopy, but in LabNotebook itself. See Print to Notebook Entry. However, the File menu Print option will continue to use your Hardcopy setting.

From the menu bar, choose **Utilities > Utilities Setup > Hardcopy** to display the Hardcopy dialog.



Send to Printer

Add Printer

NOTE: Any printer compatible with the oscilloscope's Windows OS is supported. Minimize the oscilloscope application and use the Windows controls to install printer drivers. Connect printers via LAN (Ethernet) or USB.

1. On the Utilities **Hardcopy** dialog, choose **Printer**.
2. Touch the **Add Printer** button that appears. A Microsoft Windows Devices and Printers window opens where you can configure a new printer.
3. To make the printer the instrument default, select it from the **Select Printer** list.

PRINT SETUP

1. On the Hardcopy dialog, choose **Printer**.
2. Touch **Select Printer** and choose a printer from the list. If you don't see the printer you want, first follow steps to Add Printer.
3. Choose a page **Orientation**: portrait or landscape.
4. Optionally, choose a color scheme and hardcopy (print) area.
5. Optionally, touch **Properties** to open the Windows print dialog and adjust printer properties.

Print to File

Image files can be saved to any folder on the oscilloscope hard drive, or to an external drive connected to a USB port.

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 print filenames, until you change it.

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

4. Optionally, enter the path to a new save **Directory**, or touch the **Browse** button and navigate to the folder.

   NOTE: The default print folder is C:\...\XStream\Hardcopy. Other types of files that may be saved using other oscilloscope functions, such as masks and scripts, have their own XStream subfolders.

5. Optionally, choose a color scheme and hardcopy (print) area.

Copy to Clipboard

This procedure copies the screen to the clipboard so you can paste it into another application (Microsoft Word, for example).

1. On the Hardcopy dialog, choose **Clipboard**.
2. Optionally, choose a color scheme and hardcopy (print) area.

Send to 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 output **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. Optionally, choose a color scheme and hardcopy (print) area.
5. To go on and set up the e-mail connection, touch **Configure E-Mail Server and recipient**.

Choose Print Color Scheme

To change the color of your print output, touch the **Color** button on the Hardcopy dialog and choose from:

- **Standard**(default) - prints objects on a black background, as they appear on the display.
- **Print** - prints objects on a white background using your chosen colors. This option saves ink.
- **Black & White** - prints objects in grayscale.

Set Print Area

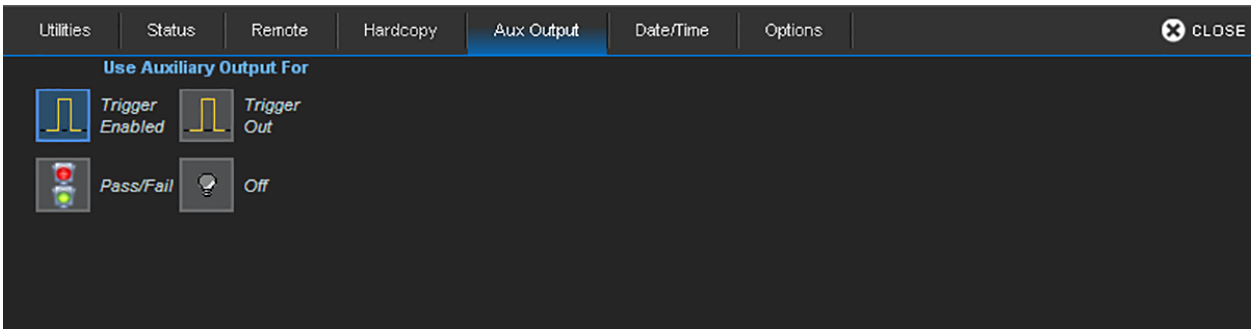
To limit which part of the touch screen is captured, touch **Hardcopy Area** on the Hardcopy dialog and choose from:

- **Grid Area Only** - omits dialogs and menus and prints only the grids.
- **DSO Window** - prints the dialogs with the grids.
- **Full Screen** - prints the entire touch screen.

Auxiliary Output Settings

Use the Aux Output dialog to configure the output of the **Aux Out** and **Cal Out** ports.

Use the Aux Output dialog to configure the output of the **Aux Out** port. The port outputs a 1.0 V TTL pulse following the selected event.



- **Trigger Enabled** sends a pulse when the oscilloscope trigger is ready (Ready indicator lit), but not necessarily fired. It can be used as a gating function to trigger another instrument when the oscilloscope is ready.
- **Trigger Out** sends a pulse when the oscilloscope trigger fires (Trig'd Indicator lit). It can be used to trigger an external oscilloscope off the instrument's state.
- **Pass/Fail** generates a pulse when Pass/Fail testing is active and conditions are met. With this selection, a **Pulse Duration** data entry control appears. Provide a value within your instrument's specified range, which varies by model. Refer to datasheet specifications at teledynelecroy.com.

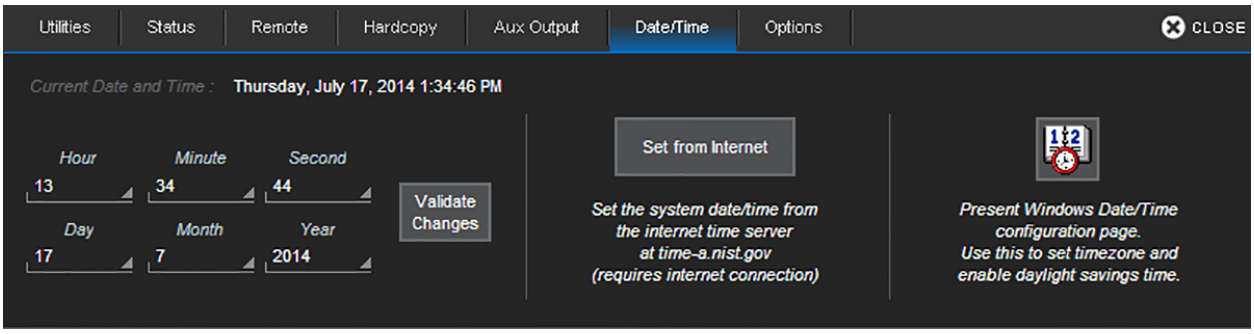
Off disables auxiliary input/output.

Date/Time Settings

Date/Time settings control the oscilloscope's timestamp. These numbers appear in the oscilloscope message bar and on tables/records internal to the oscilloscope application, such as History Mode and WaveScan.

NOTE: This is not the same as the Timebase reference clock used to synchronize traces.

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 oscilloscope must be connected to an internet access device through the LAN (Ethernet) port on your instrument's I/O panel. .

If your connection is active, touch the **Set from Internet** button.

Windows Method

To set date and time using the internal Windows system clock, touch the **Windows Date/Time** button. This displays the standard Windows **DateTime Properties** pop-up dialog, where you can further configure these settings. If you are satisfied with the setup, just touch OK.

Options

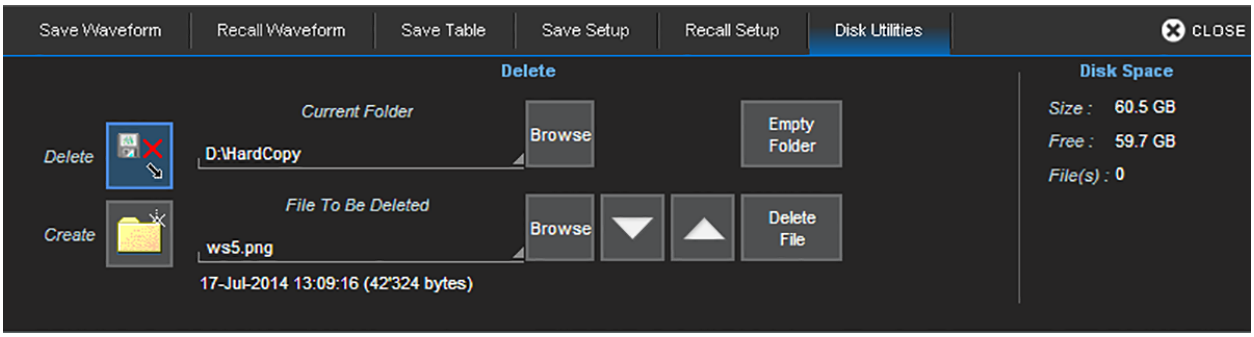
The **Options** dialog is used to add or remove software options. This dialog also displays the **ScopeID** and **Serial #**. See Adding an Option Key for instructions on using this dialog.

Disk Utilities

Use the Disk Utilities dialog to manage files and folders on your instrument's hard drive. Disk Space information is shown at the far right of the dialog for convenience.

NOTE: These tasks can also be accomplished using the standard Microsoft Windows file management tools. Choose **File > Minimize** to access the Windows desktop and task bar.

Access the **Disk Utilities** dialog by selecting **Utilities > Disk Utilities** from the menu bar, or choose any of the Save/Recall functions and open the Disk Utilities tab.



Delete a Single File

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

Delete All Files in a Folder

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

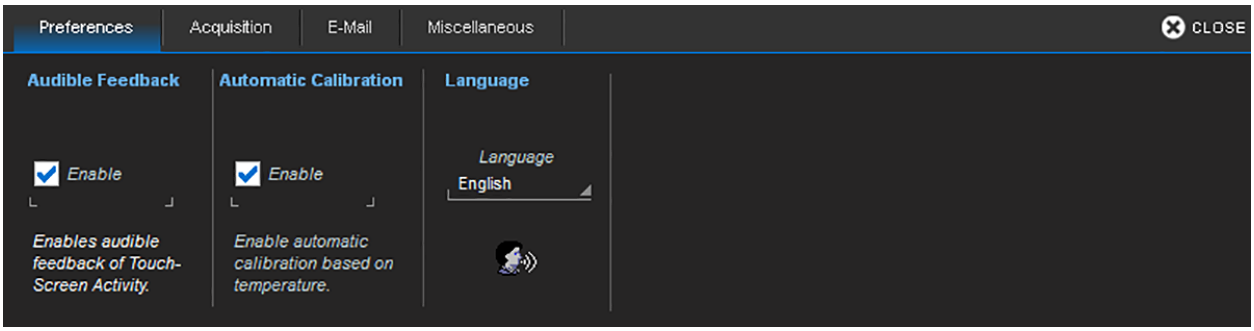
Create a New Folder

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

Preferences Settings

Preference settings have mostly to do with the appearance and performance of the oscilloscope itself, rather than the oscilloscope's interaction with other devices/systems.

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



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.

Automatic Calibration enables or disables the temperature dependent calibration feature. When enabled, the instrument will offer you a choice of calibrations to perform whenever there is a significant change in ambient temperature.

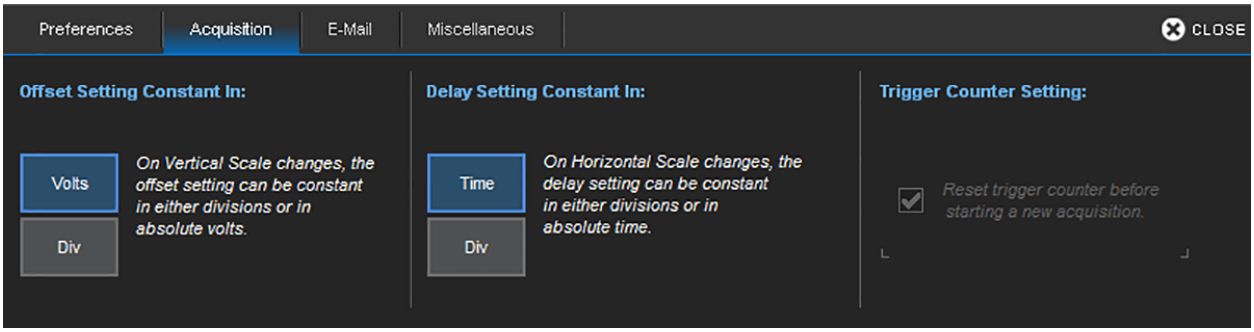
NOTE: If you do not enable this option, the oscilloscope re-calibrates only at startup and whenever you change certain operating conditions.

Language sets the language used on the display.

There are also tabs linking to Acquisition, E-Mail, and Miscellaneous settings.

Acquisition Settings

The Acquisition settings determine how traces behave on screen as gain or timebase changes.



Offset Setting constant in:

- **Volts** moves the vertical offset level indicator with the actual voltage level.
- **Div**(isions) keeps the vertical offset level indicator stationary. The waveform remains on the grid as you increase the gain; whereas, if Volts is selected, the waveform could move off the grid.

Delay Setting constant in:

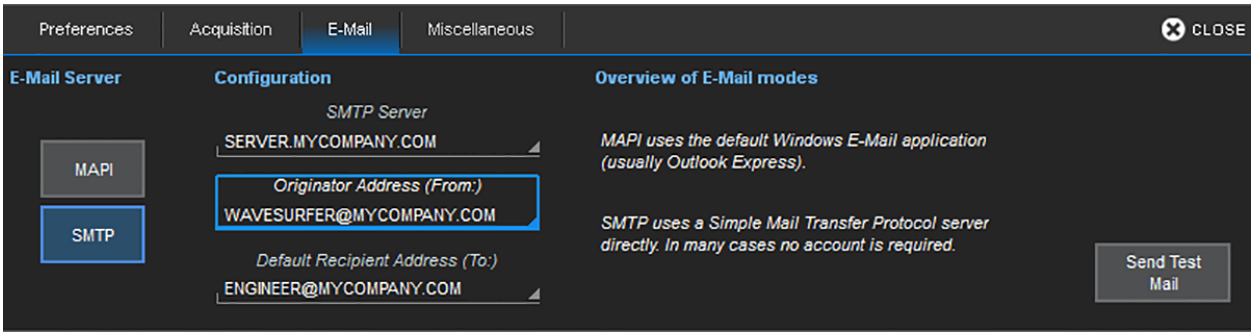
- **Time** moves the horizontal offset level indicator with the trigger point.
- **Div**(isions) keeps the horizontal offset indicator stationary. The trigger point remains on the grid as you increase the timebase; whereas, if Time is selected, the trigger point could move off the grid.

NOTE: The Offset is always in volts, and the Delay is always in time. However, whenever Div is selected, these are scaled proportional to the change in gain or timebase, thereby keeping the division of the grid constant.

Checking **Reset trigger counter before starting a new acquisition** clears the trigger counter each time the oscilloscope issues an acquisition command. It is only available when trigger Holdoff is set.

E-Mail

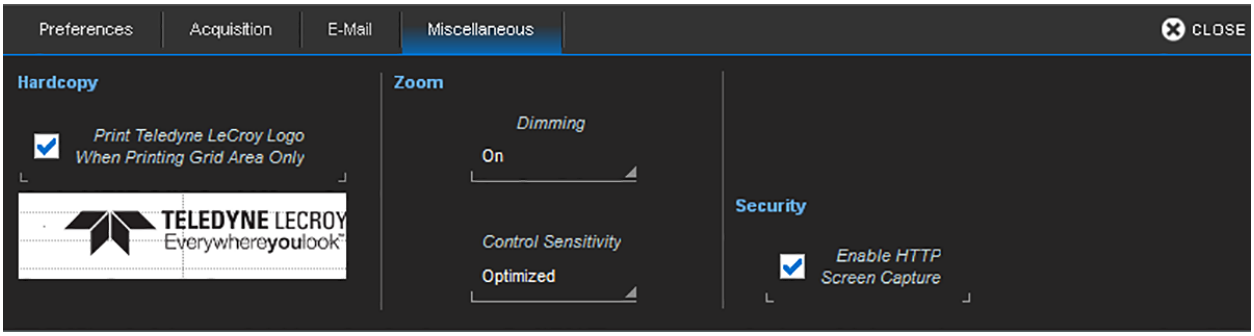
Use the E-mail dialog to set up e-mail on the oscilloscope.



1. 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.
2. Under **Configuration**:
   - **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.
3. Touch **Send Test Mail** to send a confirmation message to ensure proper e-mail configuration.

Miscellaneous Settings

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.

You can adjust zoom behavior as follows:

- **Dimming** darkens/shades those areas of the source waveform that are not part of the Zoom trace.
- **Control Sensitivity** adjusts the sensitivity of the front panel knobs. **Optimized** applies an acceleration algorithm to the knobs. **Legacy** detects rotation of the front panel knobs in a manner similar to our legacy oscilloscopes.

Serial Decode Annotation Position: If you have Serial Trigger or Decode options installed on your oscilloscope, this control determines the placement of annotation labels relative to the trace line. It does not appear if there are no installed options.

- **On Trace** places the label close to the line.
- **On Noisy Trace** sets the label further from the line to accommodate potential noise spikes in the trace.

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

Save/Recall

The oscilloscope **File menu** allows you to save or retrieve waveform files, measurement table data, and instrument setup panels.

Access these functions by choosing any of the Save or Recall options from the File menu. The dialog contains a tab for each file function.

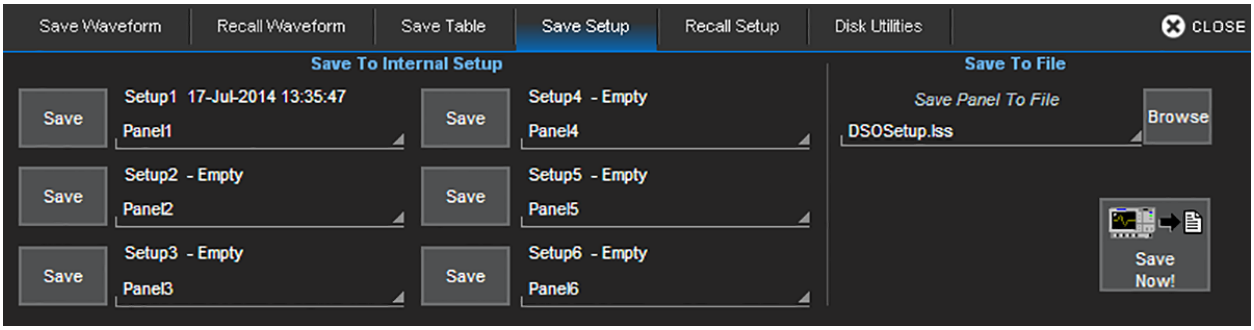
Save/Recall Setups

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

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

Saving Oscilloscope Setups

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.

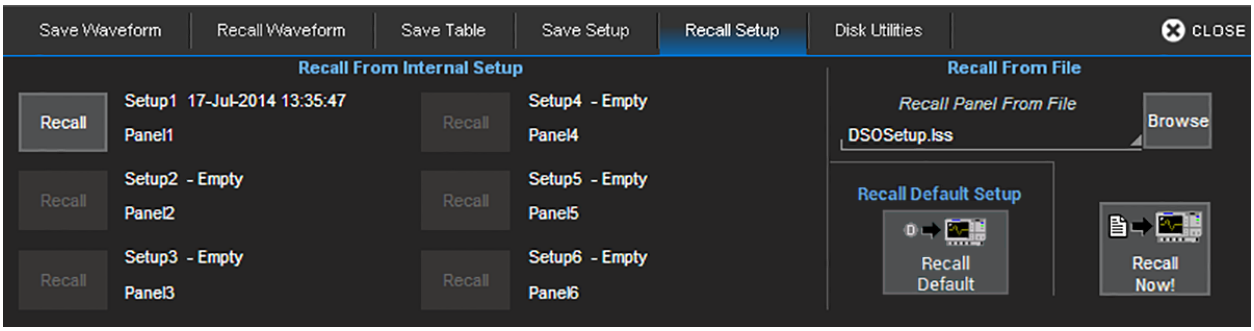
The save date/time is displayed above the **Setup** data entry control.

Save Setup To File

1. In **Save Panel to File**, touch **Browse** and navigate to the desired folder.
2. Enter a **File name**, or choose a existing file to overwrite. Touch **OK**.
3. On the Save Setups dialog, touch **Save Now!**

Recalling Oscilloscope Setups

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/Recall Waveforms

The Save Waveform function saves trace data to either an internal memory location, or to a text or binary format file (.trc). The source waveform can be any trace; for example, a channel, math function, or a waveform stored in another memory.

By default, trace files are saved to the , 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).

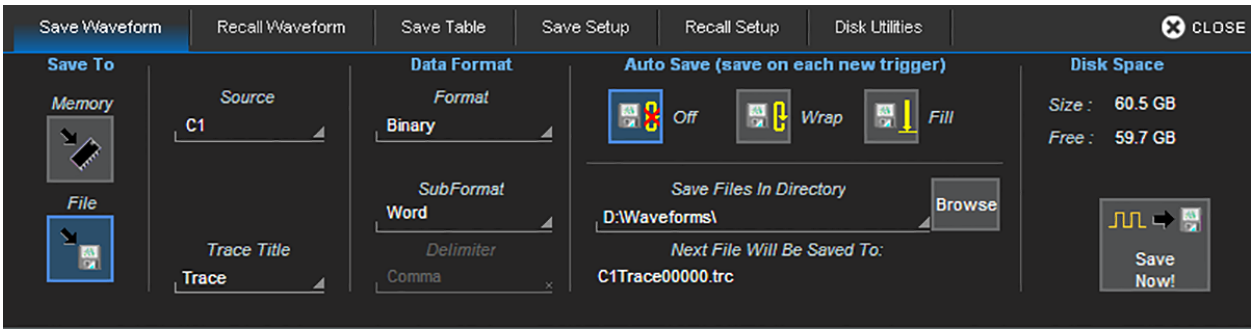
Use Recall Waveform to restore these previously saved waveforms to the display.

NOTE: Only .trc files saved in binary format can be recalled into the oscilloscope.

Save Waveform

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

NOTE: This dialog is for saving waveforms. Save S-parameter files using the Result Actions Dialog.



SAVE WAVEFORM TO MEMORY

1. Touch **Memory**.

   NOTE: When Memory is selected, only Source and Destination controls are shown on the Save Waveform dialog. When File is selected, many more controls are available.

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 at the end of this name are truncated because the instrument appends a

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.

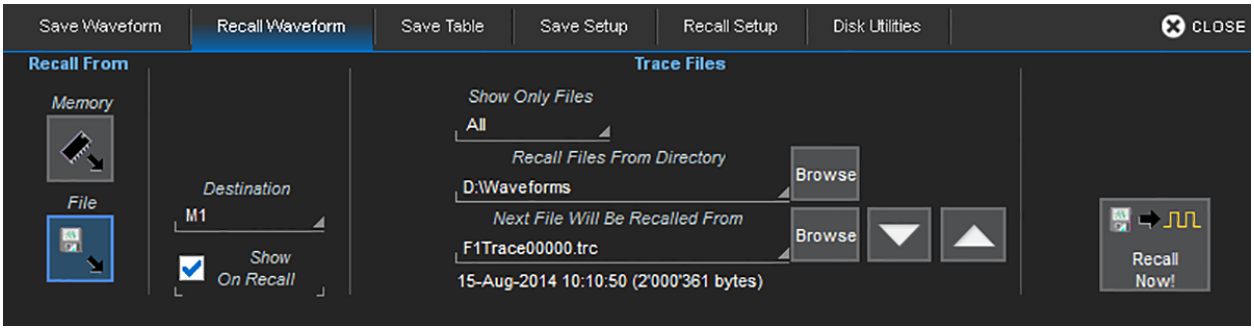
     NOTE: Binary files can be converted to ASCII using Teledye LeCroy utilities such as ScopeExplorer or WaveStudio.
   - **ASCII** text file (.txt extension).
   - **MATLAB** text file (.dat extension).
   - **Excel** text file (.csv extension).
   - **MathCad** text file (.prn extension).
   - **Audio** .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. Touch **OK**.

   NOTE: If the oscilloscope 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 oscilloscope must have access to this directory.
8. On the Save Waveform dialog, touch **Save Now!**

Recall Waveform

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

NOTE: Only .trc files saved in binary format can be recalled into the oscilloscope.



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

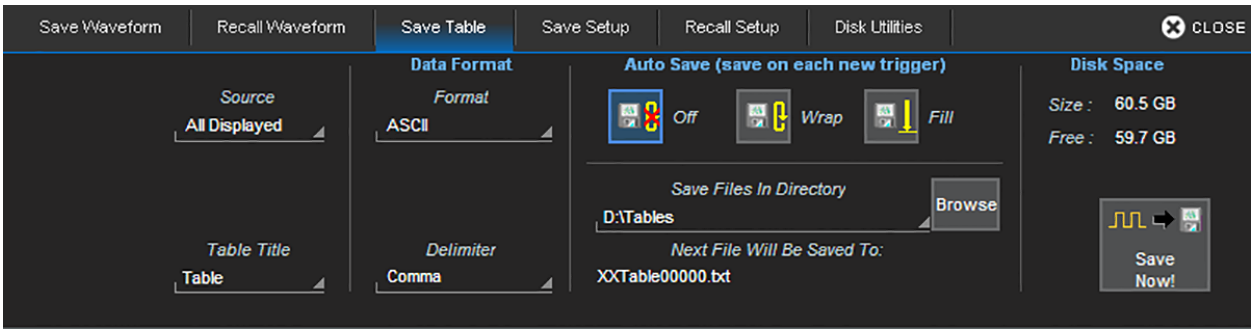
1. Touch **File**.
2. Touch **Recall files from directory** and enter the path to the waveform folder, or touch **Browse** and navigate to the folder.
3. Use the **Up /Down Arrows** to cycle through the available files until the desired file is selected.

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

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 at the end of this name are truncated because the instrument appends a 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. Select it and touch **OK**.

   NOTE: If the oscilloscope 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 oscilloscope must have access to this directory.

6. On the Save Table dialog, touch **Save Now!**

LabNotebook

Teledyne LeCroy's LabNotebook feature extends the documentation capabilities of your oscilloscope. It allows you to create and save Notebook Entries containing all displayed waveforms, the oscilloscope setup under which they were taken, and custom annotations.

Notebook Entries are stored in an internal database and are available for recall at any time. Besides storing the waveform data, LabNotebook also stores your panel setups and parameter measurements. You can back up this database to external media for indefinite storage of waveform data.

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

Create Notebook Entry

A Notebook Entry is a snapshot of the oscilloscope 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 database of My Notebook Entries accessible from the LabNotebook dialog, where 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. You can leave this as is, append some descriptive text to it, or completely remove it from your title.

   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.
4. Touch **Save**.
5. Use the Drawing toolbar to annotate the Notebook Entry. Click **Done** when finished.

LabNotebook Drawing Toolbar

The basic Notebook Entry is a screen capture of the display showing the grids as they were at the time it was taken. When an entry is first captured, it is immediately displayed in the Drawing window for you to annotate.

A variety of markup tools are available from the toolbar along the top of the window. To use any tool, touch the icon, then touch the point on the image where you wish to draw or add text.

From left to right, the tools are:

Pen Tool enables you to draw in freehand. Maintain contact with the screen to make a continuous mark. Once you release, you can touch-and-drag the object to any point on the image.

Circle Tool draws a circle around a waveform feature that you want to emphasize. Touch-and-drag across the diameter of the circle. When you release, the circle is placed. You can drag the circle to any location on the image.

Arrow Tool draws lines with arrowheads for placing callouts. You can rotate these lines through 360 degrees or drag them to any location on the image.

Text Tool opens a textbox for placing labels/annotations on the image. Touch the point on the image to place the label, then enter the text in the pop-up dialog. Once placed, you can resize the textbox or drag it to any location on the image.

Red, Green, and Blue **Color Selectors** let you quickly change the markup color. Just touch the icon, then choose the next drawing tool.

To use additional colors, touch the **More Button**. This activates a **Custom** color field. The default custom color is Yellow. To choose another, touch the color swatch, then select from the Color dialog (the standard Windows Palette dialog). You can enter RGB values, or choose from the spectrum. After saving, the new color appears in the Custom field. This remains the markup color until you choose another.

Erase Buttons allow you to remove all or selected drawing objects. Erase All will also undo any Custom color selection.

Undo Button cancels the last action. Use it to restore any objects you inadvertently erased.

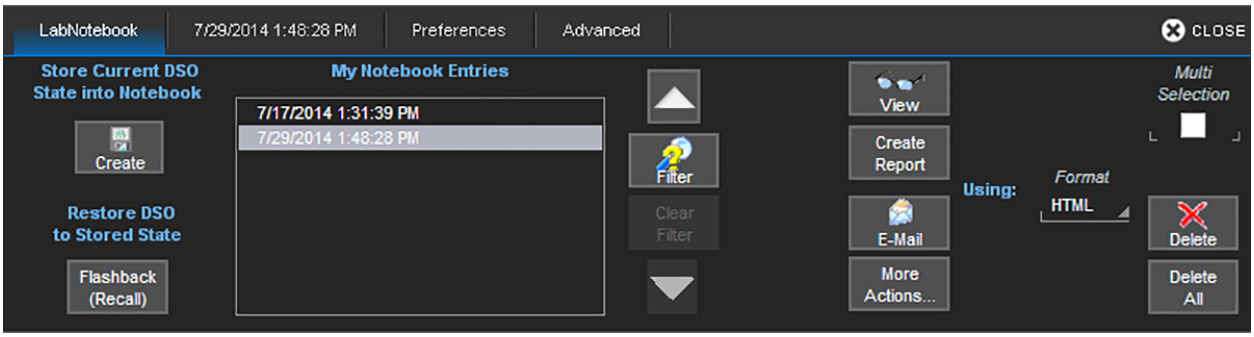
Move Toolbar Button undocks the drawing toolbar so you can move it anywhere on the display. This helps to keep tools handy when working on a particular area of a waveform. Touch the button again to restore the toolbar to the top of the Drawing window.

Done Button saves the annotations with the image and closes the Drawing window.

Manage Notebook Entries

The LabNotebook dialog is the principal notebook management tool where you can filter, select, view, edit, print, email, save, export/import, or recall Notebook Entries created in the course of your work.

To access the LabNotebook dialog, choose **File > LabNotebook** from the menu bar.



NOTE: If an external monitor is connected, LabNotebook automatically opens on the external monitor.

Select Notebook Entries

You must select Notebook Entries before any further action can be performed.

1. Select the notebook from the **My Notebooks** list.
2. Use the **Up** and **Down arrows** to scroll the My Notebook Entries list. The selected entry is highlighted in blue.

   OR

   To select multiple entries, first check Multi-selection, then Select All or scroll the list touching Select as a desired entry is highlighted. A white arrow appears next to each selected entry.
3. To remove selections from the list, highlight them again and touch **Clear**, or just touch **Clear All**.

Filter Notebook Entries

If there are a large number of notebook entries, you can apply filters to the list before selecting.

1. Select the notebook from the **My Notebooks** list.
2. Touch the **Filter** button.
3. On the **Filter Entries** pop-up, enter the filter criteria. You can use **Day/Month/Year**, a **Keyword**, or a combination.
4. Touch **Find Now** to filter.
5. To restore the full list, touch **Clear Filter**.

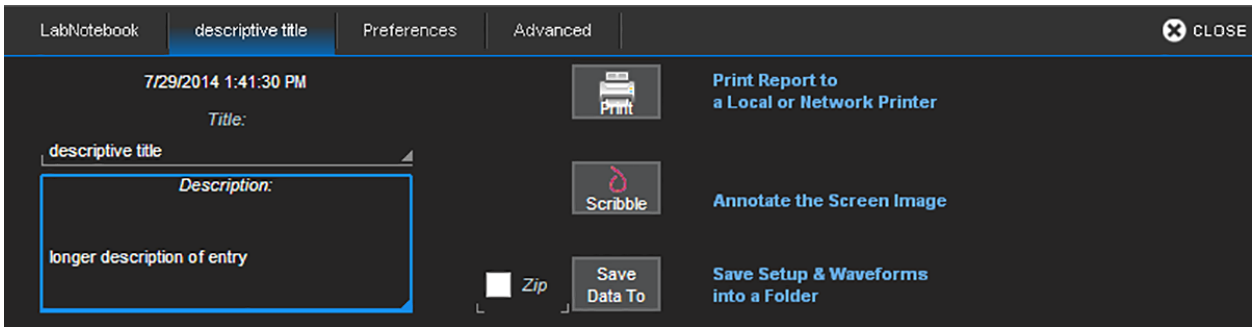
View Notebook Entries

View allows you to preview the selected entries in the report format before printing/saving.

Select the desired entries and touch the **View** button. Use the scrollbar that appears on the LabNotebook window to navigate the report.

Edit Notebook Entries

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



4. Modify the **Title** or **Description**.
5. To add markup to the entry, touch the **Scribble** button and use the Drawing Toolbar.

Email Notebook Entries

Choose **E-Mail** to send selected Notebook Entries to the default address specified in the oscilloscope Preferences. To use the E-Mail button, the instrument must have an active network connection and you must first configure the email address and server.

If you have not yet configured email, or if you wish to change the recipient address before sending, open the LabNotebook Preferences tab, then touch the **Configure E-Mail** button.

Also select whether or not to **Attach Setup & Waveform** files to the email with the LabNotebook files.

Print Notebook Entries

To print multiple entries, select them on the main LabNotebook dialog, then touch the **Print button** on the same dialog.

To print a single entry, select it on the main LabNotebook dialog, then go to the **second tab** and touch the **Print button**.

Delete Notebook Entries

Use the **Delete** button to remove selected Notebook Entries, or **Delete All** to clear the entire **My Notebook Entries**.

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

Create Report

Create Reports collates the selected Notebook Entries into a single .RTF/.PDF document or HTML archive using the report template selected on the LabNotebook Preferences tab. This can be one of the preformatted templates or a custom format. It is not necessary to first create a report document to view, email, or print selected Notebook Entries.

1. Select the notebook from the **My Notebooks** list.
2. Select the entries from the **My Notebook Entries** list.
3. Choose the output **Format**.
4. Touch **Create Report**.
5. On the **Create Report** window, select the folder in which to save the report.

   TIP: Touch **Open Explorer Here** and use the Windows Explorer to create a new folder. After closing the Explorer, touch the **Refresh** button to display the folder in the Create Report window.
6. Enter a **File name** for the report and click **OK**.

Print to Notebook Entry

The front panel Print button can be configured to capture the display and create a new Notebook Entry. This is a convenient way to create new Notebook Entries as you work.

To configure the Print button for Notebook Entries, go to **File > LabNotebook > Preferences tab** and check **Create Entry when Hardcopy Pressed**.

Flashback Recall

Once a Notebook Entry is made, you can recall it at any time using Flashback Recall. The recall includes waveforms and oscilloscope 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** and **Notebook Entry** from the lists.
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 and appears on reports).
- **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.

Manage Notebooks

LabNotebook stores Notebook Entries in a .zip archive on the instrument hard drive. Each .zip file is one Notebook comprised of everything shown in the My Notebook Entries list. New Notebooks can be created for different individuals or projects, or an existing Notebook backed up for storage.

NOTE: The default Notebook is D:\Xport\MyLabNotebook.zip. If you've already created Notebook Entries that you wish to keep, you can use the backup feature to save them under a new file name or location before starting a new Notebook.

Create New Notebook

1. Choose **File > LabNotebook**.
2. Open the **Advanced** dialog and choose **Start New**.
3. Enter a **File Name** for the new Notebook (optionally, choose a new storage folder, as well). Touch **OK**.

The new notebook now appearsin the Database field on the Advanced dialog. New Notebook Entries will be added to this file whenever it is selected as the Database.

Back Up Notebook

1. Choose **File > LabNotebook**.
2. Open the **Advanced** dialog and choose **Backup**.
3. Optionally, enter a new **File Name** or choose a new storage **Folder**.

   NOTE: The default is the current notebook name with the extension *.bak.zip appended to it. You can change this to anything you like, although it is recommended to leave the .zip file extension.
4. Choose to **Backup to Removable Disk** (this option is active if you have a USB drive attached to the oscilloscope) or **Backup to Folder** on hard drive.

Import Notebook

Archived notebooks can be imported into LabNotebook. This notebook can not be selected as the working Database on the Advanced dialog, and the entries in it are appended to the My Notebook Entries list for selection.

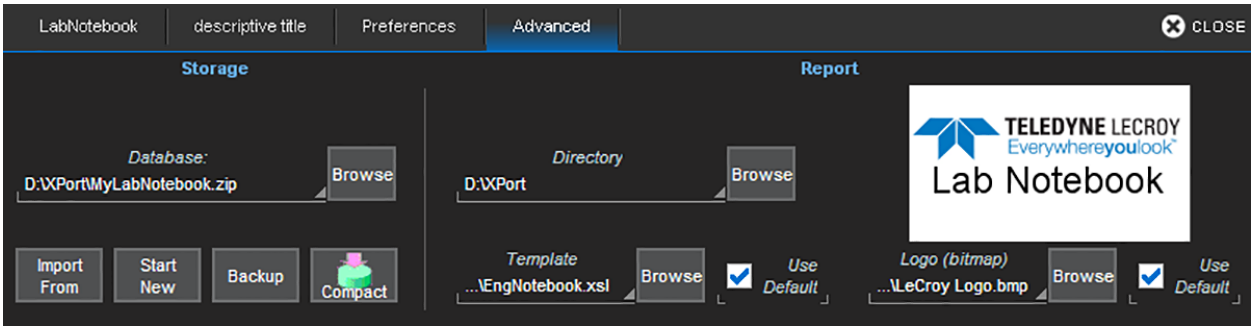
1. Choose **File > LabNotebook**.
2. Open the **Advanced** dialog and touch **Import From**.
3. Navigate to the desired archive and select it. Touch **OK**.

Delete Notebook

Use the standard Windows controls to browse to the notebook in the D:\Xport folder on the oscilloscope hard drive and delete it.

Customize Reports

The Advanced tab allows you to customize the report creation function.



Change Directories

To change where Notebooks are stored, change the **Notebooks Directory** folder.

To change where reports are output, change the **Report Directory** folder. You can choose an external location, such as a USB drive.

Change Report Template

Deselect **Use Default** next to the Template field, then touch **Browse** and select a different template from the D:\Xport folder.

You can create your own report template and place it in this directory for selection. Templates must be saved as .xsl or .xslt files.

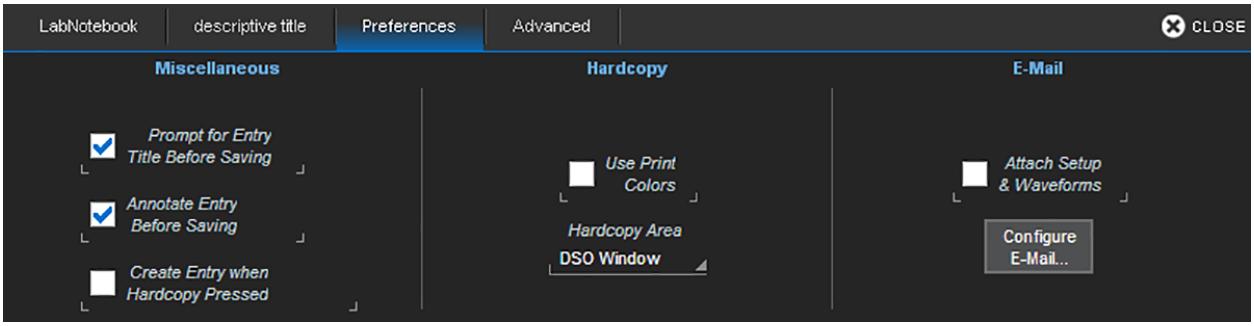
Change Logo

The included LabNotebook report templates use our logo as a placeholder. You can replace this with your custom logo. Logo files should be in bitmap (.bmp) format and not exceed 100 pixels high by 180 pixels wide.

1. Copy the logo file to the D:\Xport folder.
2. Choose **File > LabNotebook**, then touch the **Advanced** tab.
3. Deselect the **Use Default** checkbox next to the Logo field.
4. Touch the **Browse** button and navigate to the the new logo file. Select and touch **OK**.The new file path appears in the Logo field with a preview of the image above it.

Configure LabNotebook Preferences

To configure the behavior of the LabNotebook tool, on the menu bar, choose **File > Lab Notebook**, then touch the **Preferences** tab.



Select/deselect the following options:

Prompt for Entry Title Before Saving opens the LabNotebook dialog when a new entry is created. You can elect to name notebook entries using only the date/timestamp by leaving this box unchecked.

Annotate Entry Before Saving opens the Drawing Toolbar to annotate a notebook entry as soon as it is created.

Create Entry When Hardcopy Pressed configures the front panel print button to create a new notebook entry whenever it is pressed.

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

Hardcopy Area determines how much of the screen image is included in the report: grid area only, grid area plus dialog, whole screen. Touch the field and choose from the pop-up menu.

Attach Setup & Waveforms attaches these files for each trace in the report: waveform data (.trc), a screen dump (.png), oscilloscope setup file (.lss), report template file (.xsl), and export record (.htm).

Optionally, touch the **Configure E-Mail** button to set the recipient address and server information on the Preferences E-mail dialog.

Maintenance

Cleaning

Clean only the exterior of the instrument using a soft cloth moistened with water or an alcohol solution. Do not use harsh chemicals or abrasive elements. Under no circumstances submerge the instrument or allow moisture to penetrate it. Avoid electric shock by unplugging the power cord from the AC outlet before cleaning.

CAUTION. Do not attempt to clean internal parts. Refer to qualified service personnel.

Calibration

The oscilloscope is calibrated at the factory prior to being shipped. This calibration is run at 23° C (± 2° C) and is valid for temperatures ± 5° C of the original calibration temperature. Within this temperature range the oscilloscope will meet all of the specifications.

The oscilloscope will offer you two calibration options whenever the temperature ranges outside this limit:

- **Calibrate All**possible combinations of vertical and horizontal settings at the current temperature. This calibration is valid for the current temperature ± 5° C and takes about minutes.
- **Calibrate Current Setting** (vertical and horizontal). This calibration is valid for only these settings at the current temperature ± 5° C and takes under 30 seconds.

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

Schedule an annual factory calibration as part of your regular maintenance. Contact us about extended warranty, calibration, and upgrade plans available for purchase.

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.

Reboot Oscilloscope

To restart the oscilloscope application, choose **File > Exit** 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 oscilloscope, which includes restarting the OS:

1. Shut down the instrument by choosing **File > Shutdown**.
2. Wait 10 seconds, then press the **Power button** on the front of the oscilloscope.

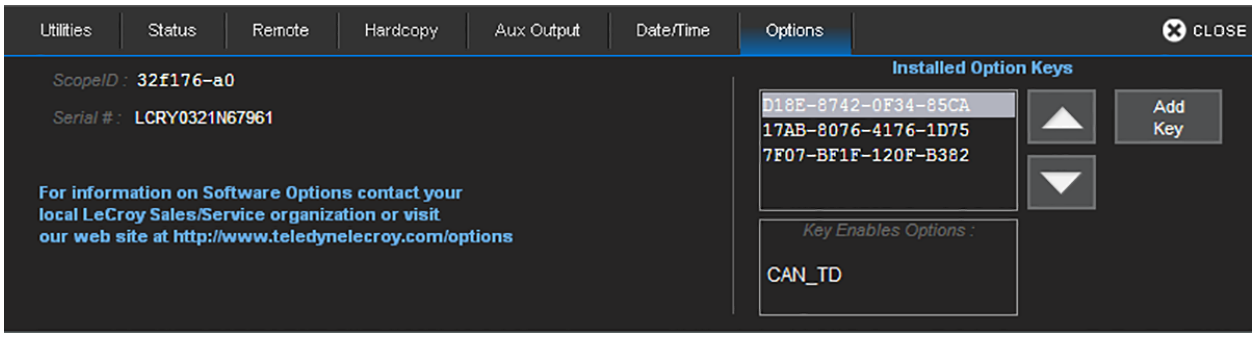
Adding an Option Key

Many optional software packages are available to extend the Analysis functions of the oscilloscope. See the product page at teledynelecroy.com for a list of options compatible with your model.

Contact your local Teledyne LeCroy representative or national distributor to purchase an option. You will receive a Key Code by email that enables the new functionality.

To install the key and activate the software:

1. From the menu bar, choose **Utillities > Utilities Setup**, then touch 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 **O.K.** on the keyboard to enter the information.

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

4. Restart the oscilloscope application: choose **File > Shutdown**, then double-click the **Start DSO** icon on the desktop.

X-Stream 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 saved panel setups, waveforms, screen captures, calibration constants, or other data stored on the oscilloscope's D: drive.

1. Exit the oscilloscope application by choosing **File > Exit**.
2. Visit our download page at teledynelecroy.com/support/softwaredownload and click the link to **Oscilloscope Downloads > Firmware Upgrades**.
3. Select your oscilloscope **series** and **model number**.
4. Enter your registration **login** information, or create a new account.
5. Click the **download** link, and choose to **Save** the installer to the oscilloscope Desktop or a folder on the D: drive. If downloading from a remote PC, save the installer to a USB storage device to transfer it to the oscilloscope.
6. Browse to the location of the installer (xstreamdsoinstaller_x.x.x.x.exe) and double-click it to launch the X-Stream Setup wizard.
7. On the wizard, click **Next**, then read the EULA and click **I Agree**.
8. Leave the default installation (recommended), or select individual components:
   - Drivers for GPIB1 - required for internal PCI-GPIB card.
   - MATLAB MCR - required for sampling scopes, QPHY-USB and QPHY-Broad-R-Reach.
   - X-Stream DSO, DSO Device drivers, Upgrade DSO Microcode - required for a version upgrade.
   - Drivers for CAN - required for external CAN trigger module.
   - Touch Screen Driver - required to control the oscilloscope using the display as a touch screen (you can opt to use a mouse).
   - Driver for MS-32 - required for Mixed Signal device MS-32.
   - Drivers for MSxx - required for Mixed Signal devices MS-250 and MS-500.
   - LSIB Package - required for LSIB host card.
   - SPARQ Package - required to drive a connected SPARQ from the oscilloscope.

   Click **Install** when done.

   NOTE: Not all components are available on all models. If you do not see an option listed, it is not required for your oscilloscope.

9. If you receive Windows security warnings, **trust** and **Install** the file. If you see the Hardware Programmers screen, accept all code installations, then click **Close** to return to the X-Stream Setup wizard.
10. When installation is complete, choose **Reboot now** and click **Finish**.

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.**

System Recovery

Your oscilloscope is designed to operate reliably for many years. The application software operating the instrument runs on a Microsoft Windows® platform. The loading or incomplete removal of additional Windows applications can cause problems in the stability of the operating system. Severe cases may require you to reload the base operating system and oscilloscope application.

For this purpose, Teledyne LeCroy provides a recovery application and a backup image in an extra partition on the instrument's hard drive. The recovery process is easy to perform.

Since third-party recovery software is used, our instructions may not be the most definitive or current. We encourage you to take advantage of additional resources available from the vendor's website at:

acronis.com/homecomputing/download/docs/

Before You Start

- Find your Windows Product Key number, usually listed on a sticker on the back of the oscilloscope.
- If you intend to reactivate Windows through the Internet, connect a network cable to the oscilloscope's LAN port.
- Connect a keyboard and mouse via the USB host ports.

Using True Image Home

1. Apply power to the oscilloscope.
2. During the startup process, as soon as you see the message "Starting Acronis Loader... Press F11 for Acronis Startup Recovery Manager," press the **F11** key until the Acronis logo appears momentarily. The Acronis window is displayed.

   NOTE: Do not press F11 before you see the "Starting Acronis Loader..." message or you will enter the boot device selection menu. If a bootmenu dialog box appears, press Cancel or Esc.
3. Select **Acronis True Image Home (Full Version)**.
4. On the Acronis True Image Home page, under options for Recover, select **My Disks**. The Recovery Wizard opens.
5. On the Recovery Wizard, under Archive Selection, select the disk archive that has a create date, then click **Next**.

6. Under Recovery Method, select **Recover whole disks and partitions**, then click **Next**.
7. Under What to Recover, select **NTFS (SYSTEM) (C:)**, then click **Next**.
8. Under Settings of Partition C, in the top section, Partition location (required), select **New Location**.The Partition Destination window opens.
9. Under New Partition Location, select **NTFS (SYSTEM) (D:)**, then click **Accept**. This returns you to the Settings of Partition C step. Click **Next**.
10. A summary window is displayed indicating that Acronis True Image is ready to proceed with recovering partition C -> D. Click **Proceed** to start the recovery process.

    NOTE: Recovery takes approximately 4 to 15 minutes depending on the version of operating system that is being restored. The progress is displayed on the screen.
11. When recovery is complete, you will see the message "Recover operation succeeded." Click **OK**.
12. Click the **close button** to exit the Acronis window. The oscilloscope will restart and begin installing the required software.

    NOTE: If prompted to restart Windows, select Restart Later.

Reinstall the Oscilloscope Application

1. When the X-Stream DSO Setup Wizard appears, click **Next**.
2. On the License Agreement page, select **I Agree**.
3. On the Choose Components page, select the **default (installation)** and click **Install**.

   NOTE: You may see a Windows Security message indicating that Windows can't verify the publisher of the driver software. Choose "Install this driver software anyway" and click Install.
4. When the X-Stream installation is complete, choose to **Reboot now** and click **Finish**.
5. When prompted, enter your Windows Product Key number to re-activate Windows.

Reactivate the F11 Startup Utility

In order for the system recovery wizard to be accessed again from the boot menu, you must reactivate the F11 startup utility. It is critical to complete these steps after your oscilloscope has restarted.

1. From the Windows **Start Menu** choose **All Programs > Acronis > Acronis True Image Home**.
2. On the Acronis True Image Home window, towards the top right, click **Tools & Utilities**.
3. On the Tools & Utilities page, beneath Protection Tools, click **Acronis Startup Recovery Manager**.
4. On the Acronis Startup Recovery Manager window, click **Activate**. F11 boot time recovery is now enabled.

   NOTE: The process "Searching for Acronis Secure Zone..." may take several minutes.

Restore Software to Current Revision Levels

The recovery process produces a replica of the operating system and oscilloscope application software at the revision levels that were current when the oscilloscope was manufactured. It does not automatically upgrade the:

- Oscilloscope application software (X-Stream)
- Windows operating system
- Virus scan definition files
- Drivers for some hardware options and accessories

Therefore, after completing the disk image recovery, it is highly recommended to search vendor websites and upgrade the individual components to current revision levels.

The latest oscilloscope application software can be downloaded directly from the Teledyne LeCroy website at teledynelecroy.com. Most required hardware drivers can be installed during the firmware upgrade.

Since the calibration data for the oscilloscope is stored in the D: drive, current calibration constants are not overwritten during recovery of the C: drive. You do not need to restore these.

Technical Support

Phone

Registered users can contact their local Teledyne LeCroy service center at the number listed in this manual to make Technical Support requests by phone or email.

Web

Teledyne LeCroy publishes a free Technical Library on its website. Manuals, tutorials, application notes, white papers, and videos are available to help you get the most out of your Teledyne LeCroy products.

- The Datasheet published on the product page contains the detailed product specifications.
- Oscilloscope System Recovery Tools and Procedures contains instructions for using Acronis® True Image® Home included with the oscilloscope.

You can also submit Technical Support requests via the website at:

teledynelecroy.com/support/techhelp.

Returning a Product for Service

Contact your local Teledyne LeCroy service center for calibration or other service. If the product cannot be serviced on location, the service center will give you a **Return Material Authorization (RMA) code** and instruct you where to ship the product. All products returned to the factory must have an RMA.

Return shipments must be prepaid. Teledyne LeCroy cannot accept COD or Collect shipments. We recommend air-freighting. Insure the item you're returning for at least the replacement cost.

1. Remove all accessories from the device. Do not include the manual.
2. Pack the product in its case, surrounded by the original packing material (or equivalent).
3. Label the case with a tag containing:
    - The RMA
    - Name and address of the owner
    - Product model and serial number
    - Description of failure or requisite service
4. Pack the product case in a cardboard shipping box with adequate padding to avoid damage in transit.
5. Mark the outside of the box with the shipping address given to you by Teledyne LeCroy; be sure to add the following:
    - ATTN: <RMA code assigned by Teledyne LeCroy>
    - FRAGILE
6. **If returning a product to a different country:**
    - Mark the shipment as a "Return of US manufactured goods for warranty repair/recalibration."
    - If there is a cost for the service, list the cost in the Value column and the original purchase price "For insurance purposes only."
    - Be very specific about the reason for shipment. Duties may have to be paid on the value of the service.

Extended warranty, calibration, and upgrade plans are available for purchase. Contact your Teledyne LeCroy sales representative to purchase a service plan.

Contact Teledyne LeCroy

United States and Canada
- World Wide Corporate Office
Teledyne LeCroy Corporation
700 Chestnut Ridge Road
Chestnut Ridge, NY, 10977-6499, USA
Ph: 800-553-2769 / 845-425-2000
FAX: 845-578-5985
teledynelecroy.com
Support:
contact.corp@teledynelecroy.com
Sales:
customersupport@teledynelecroy.com

United States Protocol Solutions Group
Teledyne LeCroy Corporation
3385 Scott Boulevard
Santa Clara, CA, 95054, USA
FAX: 408-727-0800
teledynelecroy.com
Sales and Service:
Ph: 800-909-7211 / 408-727-6600
contact.corp@teledynelecroy.com
Support:
Ph: 800-909-7112 / 408-653-1260
psgsupport@teledynelecroy.com

European Headquarters
Teledyne LeCroy SA
4, Rue Moïse Marcinhes
Case postale 341
1217 Meyrin 1
Geneva, Switzerland
Ph: + 41 22 719 2228 / 2323 / 2277
FAX: +41 22 719 2233
contact.sa@teledynelecroy.com
applications.indirect@teledynelecroy.com
teledynelecroy.com/europe
Protocol Analyzers:
Ph: +44 12 765 03971

Singapore
Genetron Singapore Pte Ltd.
37 Kallang Pudding Road, #08-08
Tong Lee Building Block B
Singapore 349315
Ph: ++ 65 9760-4682

China
Teledyne LeCroy Corporation Beijing
Rm. 2001
Unit A, Horizon Plaza
No. 6, Zhichun Road, Haidian District
Beijing 100088, China
Ph: ++86 10 8280 0318 / 0319 / 0320
FAX:++86 10 8280 0316
Service:
Rm. 2002
Ph: ++86 10 8280 0245

Taiwan
LeColn Technology Co Ltd.
Far East Century Park, C3, 9F
No. 2, Chien-8th Road,
Chung-Ho Dist., New Taipei City, Taiwan
Ph: ++ 886 2 8226 1366
FAX: ++ 886 2 8226 1368

Korea
Teledyne LeCroy Korea
10th fl.Ildong Bldg.
968-5 Daechi-dong, Gangnam-gu
Seoul 135-280, Korea
Ph: ++ 82 2 3452 0400
FAX: ++ 82 2 3452 0490

Japan
Teledyne LeCroy Japan
Hobunsya Funchu Bldg, 3F
3-11-5, Midori-cho, Fuchu-Shi
Tokyo 183-0006, Japan
Ph: ++ 81 4 2402 9400
FAX: ++ 81 4 2402 9586
teledynelecroy.com/japan

Certifications

EMC Compliance

EC Declaration of Conformity- EMC

The oscilloscope meets intent of EC Directive 2004/108/EC 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, high quality shielded interface cables should be used.

2 Emissions which exceed the levels required by this standard may occur when the oscilloscope is connected to a test object.

3 This product is intended for use in nonresidential areas only. Use in residential areas may cause electromagnetic interference.

4 Meets Performance Criteria “B” limits of the respective standard: during the disturbance, product undergoes a temporary degradation or loss of function or performance which is self-recoverable.

5 Performance Criteria “C” applied for 70%/25 cycle voltage dips and 0%/250 cycle voltage interruption test levels per EN61000-4-11.

EUROPEAN CONTACT:

Teledyne LeCroy Europe GmbH
Waldhofer Str 104
D-69123 Heidelberg
Germany
Tel: (49) 6221 82700

Australia & New Zealand Declaration of Conformity– EMC

Oscilloscope 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 (ACMA):

EN 55011:2010 Radiated and Conducted Emissions, Group 1, Class A, in accordance with EN61326-1:2013 and EN61326-2-1:2013.

AUSTRALIA / NEW ZEALAND CONTACTS:

Vicom Australia Ltd.
1064 Centre Road
Oakleigh, South Victoria 3167
Australia

Australia Vicom New Zealand Ltd.
60 Grafton Road
Auckland
New Zealand

Safety Compliance

EC Declaration of Conformity– Low Voltage

The oscilloscope meets intent of EC Directive 2006/95/EC 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 of the instrument has been verified to conform to the following limits put forth by these standards:

- Mains Supply Connector: CAT II, local distribution level, equipment connected to the mains supply (AC power source).
- Measuring Terminals: CAT O, signal level, equipment measuring terminals connected to source circuits where measures are taken to limit transient voltages to an appropriately low level.
- Unit: Pollution Degree 2, operating environment where normally only dry, non-conductive pollution occurs. Conductivity caused by temporary condensation should be expected.
- Unit: Protection Class I, grounded equipment, in which protection against electric shock is achieved by Basic Insulation and a connection to the protective ground conductor in the building wiring.

U.S. Nationally Recognized Agency Certification

The oscilloscope 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 oscilloscope 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 2002/96/EC and 2006/66/EC on Waste Electrical and Electronic Equipment (WEEE) and Batteries.

The instrument is subject to disposal and recycling regulations that vary by country and region. Many countries prohibit the disposal of waste electronic equipment in standard waste receptacles. For more information about proper disposal and recycling of your Teledyne LeCroy product, please visit teledynelecroy.com/recycle.

Restriction of Hazardous Substances (RoHS)

This product and its accessories conform to the 2011/65/EU RoHS2 Directive, as it is classified as Industrial Monitoring and Control Equipment (per Article 3, Paragraph 24) and is exempt from RoHS compliance until 22 July 2017 (per Article 4, Paragraph 3).

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 oscilloscope’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 oscilloscope. 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.

Windows License Agreement

The WaveSurfer 10Oscilloscope software runs on a Windows® operating system. Teledyne LeCroy's agreement with Microsoft® prohibits users from running software that is not relevant to measuring, analyzing, or documenting waveforms on Teledyne LeCroy oscilloscopes.

700 Chestnut Ridge Road
Chestnut Ridge, NY 10977
USA

teledynelecroy.com