TELEDYNE LECROY
Everywhereyoulook™



Operator's Manual

HRO / WaveRunner 6 Zi Oscilloscopes

TELEDYNE LECROY
Everywhereyoulook™

WaveRunner 6 Zi/HRO Oscilloscopes Operator's Manual

© 2017 Teledyne LeCroy, Inc. All rights reserved.

Unauthorized duplication of Teledyne LeCroy, Inc. documentation materials other than for internal sales and distribution purposes is strictly prohibited. However, clients are encouraged to duplicate and distribute Teledyne LeCroy, Inc. documentation for their own internal educational purposes.

and Teledyne LeCroy, Inc. are trademarks of Teledyne LeCroy, Inc., Inc. Other product or brand names are trademarks or requested trademarks of their respective holders. Information in this publication supersedes all earlier versions. Specifications are subject to change without notice.

928453 Rev B
May 2017

Contents

Safety 1
- Symbols 1
- Precautions 1
- Operating Environment 2
- Cooling 2
- Calibration 2
- Cleaning 2
- Power 3

Oscilloscope Overview 5
- Front Panel 5
- Rotating and Tilting the Display 11
- Signal Interfaces 12

Oscilloscope Set Up 15
- Powering On/Off 15
- Software Activation 15
- Connecting to Other Devices/Systems 16
- Removable Hard Drive 17
- Language Selection 19

Using MAUI 21
- Touch Screen 21
- Touch Actions 28
- Working With Traces 30
- Zooming 34
- Print/Screen Capture 38

Acquisition 39
- Auto Setup 39
- Vertical 40
- Digital (Mixed Signal) 45
- Timebase 48
- Trigger 55
- Viewing Status 70

Display 71
- Display Set Up 72
- Persistence Display 74

Math and Measure 77
Cursors 77
Measure 80
Math 103
Memory 120
Analysis Tools 122
WaveScan 122
Pass/Fail Testing 128
Saving Data (File Functions) 133
Save 133
Auto Save 138
Recall 139
LabNotebook 141
Report Generator 147
Share 148
Print 149
Email & Report Settings 150
Using the File Browser 151
Utilities 153
Utilities Dialog 153
Status 153
Remote Control 154
Auxiliary Output 156
Date/Time 157
Options 158
Disk Utilities 159
Preferences Settings 160
Acquisition 161
Color 162
Miscellaneous 163
Maintenance 165
Touch Screen Calibration 165
Restart/Reboot Instrument 165
Firmware Update 166
Technical Support 167
Returning a Product for Service 168

Certifications 169
EMC Compliance 169
Safety Compliance 170
Environmental Compliance 171
ISO Certification 171
Warranty 172
Intellectual Property 172
Windows License Agreement 172
Index 173

Welcome

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

Take a moment to 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.

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

Sincerely,



David C. Graef

Vice President and General Manager, Oscilloscopes
Teledyne LeCroy

Safety

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

Symbols

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

 **Caution** of potential damage to instrument or **Warning** of potential bodily injury. Do not proceed until the information is fully understood and conditions are met.

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

 **Caution**, contains parts/assemblies susceptible to damage by Electrostatic Discharge (ESD).

 Frame or chassis terminal (ground connection).

 Alternating current.

 Standby power (front of instrument).

Precautions

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

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

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

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

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

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

Keep product surfaces clean and dry. See Cleaning.

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

Exercise care when lifting.

Do not operate with suspected failures. Do not use the product if any part is damaged. Obviously incorrect measurement behaviors (such as failure to calibrate) might indicate hazardous live electrical quantities. Cease operation immediately and secure the instrument from inadvertent use.

Operating Environment

Temperature: 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 3,048 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. In a benchtop configuration, leave a minimum of 15 cm (6 inches) around the sides between the instrument and the nearest object. The feet provide adequate bottom clearance. Follow rackmount instructions for proper rack spacing.



Caution: Do not block the cooling vents.

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

Calibration

Calibration ensures that the output from the analog-to-digital converters (ADCs) accurately represents the input. The instrument is calibrated at the factory prior to shipment. So that it maintains specified performance, it is factory set to perform an automatic calibration routine upon start-up and whenever conditions warrant it.

Warm up the oscilloscope for at least 20 minutes prior to use to give the instrument time to come to a stable operating temperature and complete the automatic calibration procedure. You will see a warning message on the Calibration dialog when the oscilloscope is still in the warm-up phase.

Cleaning

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



Caution: Unplug the power cord from the AC inlet before cleaning to avoid electric shock. Do not attempt to clean internal parts. Refer all maintenance to qualified service personnel.

Power

AC Power

100–240 VAC ±10% at 45–66 Hz; 100–120 VAC ±10% at 380–420 Hz; Automatic AC Voltage Selection

Power Consumption

Maximum power consumption (all accessories installed):

500 W (500 VA) for WaveRunner models
425 W (425 VA) for HRO models

Power consumption in Standby mode: 15 W

Ground

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



Caution: Use only the power cord provided with your instrument. Interrupting the protective conductor (inside or outside the case), or disconnecting the safety ground terminal, creates a hazardous situation. Intentional interruption is prohibited.

Oscilloscope Overview

Front Panel

Most of the front panel controls duplicate functionality available through the touch screen display.

All the knobs on the front panel function one way if turned and another if pushed like a button. The push action is preceded by the word, "Push."

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.



WaveRunner front panel.

Front Panel 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 50% of the input signal.

READY Indicator is lit when the trigger is armed. TRIG'D is lit momentarily when a trigger occurs.

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

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

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

Setup corresponds to the menu selection Trigger > Trigger Setup or touching the Trigger Descriptor Box. Press it once to open the Trigger setup dialog.

Trigger Scan opens the TriggerScan dialog.

The Source, Slope, and Coupling buttons cycle through the available selections. Continue pressing until the desired selection is lit.

Front Panel 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 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. Push the knob to toggle between variable (fine) and standard (1, 2, 5, 10 step) increments.

Front Panel Horizontal Controls

Turn the Delay knob to change the Trigger Delay value (S). Push the knob to reset Delay to zero.

If the trace source is an input channel, turn the Horizontal Adjust knob to set the Time/division (S) of the oscilloscope acquisition system. The value is shown on the Timebase descriptor box. When using this control, the oscilloscope allocates memory as needed to maintain the highest sample rate possible for the timebase setting.

If the trace source is a zoom, memory or math function, turning the Horizontal Adjust knob changes the horizontal position of the trace. The value is shown on the corresponding descriptor box.

Push the knob to toggle between variable (fine) and standard (1, 2, 5, 10 step) increments.

Math, Zoom, and Memory Controls

When pressed, Zoom creates a zoom trace of all visible channel traces at x10 scale of the original. Press Zoom again to turn off the zoom traces.

Pressing Math opens the math setup dialogs. By default, a new math trace is created using the active channel trace as a source. Modify the math formula on the Function (F*x*) dialog.

Pressing Mem opens the Memory setup dialog.

Miscellaneous Controls

Touch Screen enables/disables touch screen functionality. You can still operate the oscilloscope using front panel controls.

Scope Setup opens the setup flyout menu on the first press. Select Auto Setup for a full setup of all four channels; select Default Setup to reset the oscilloscope to the factory default configuration. To perform a quick setup for one channel only, select one of the Channel <#> Find Scale options. To repeat your last Auto Setup selection, you can skip the flyout menu and just press the Scope Setup front panel button again.

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

Clear Sweeps resets the acquisition counter and any cumulative measurements, persistence trace displays, statistics, histicons, and averaging.

WaveStream Indicator lights when the oscilloscope is in WaveStream mode.

Pressing the Intensity knob toggles between Analog Persistence and WaveStream Persistence modes. Turning the knob raises/lowers the intensity level, which affects the persistence display.

WavePilot Controls

The WavePilot with the SuperKnob provides quick, manual control of commonly used oscilloscope functions.

Each of the WavePilot buttons puts the SuperKnob into a different operating "mode." When pressed, a flyout menu appears to the right side of the touch screen showing the actions that may now be performed using the SuperKnob.

The SuperKnob acts like a joystick to control other objects on the touch screen. Depending on the primary mode, you may:

- Scroll waveforms and tables or make selections by shifting the knob left/right or up/down.
- Toggle the behavior of other controls by pressing the knob.
- Adjust values or rescale traces by turning the knob.

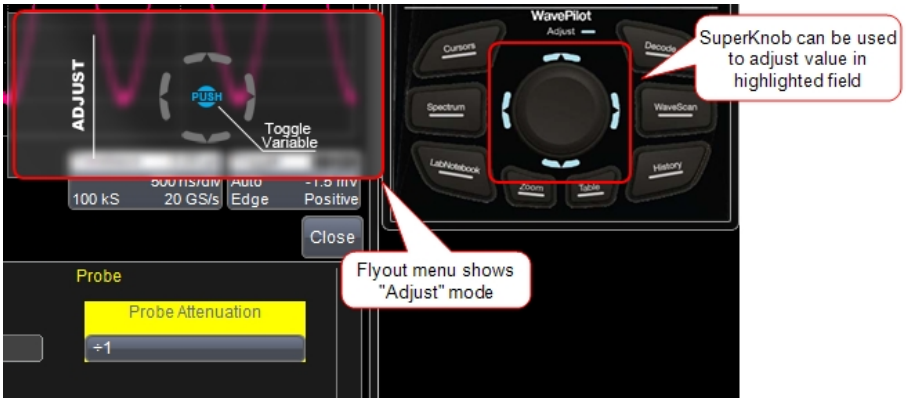
Adjust

By itself (no other WavePilot "mode" button selected), the SuperKnob acts as the Adjust knob, allowing you to modify the values in numerical entry fields.

Tip: When in this mode, the LED next to the word "Adjust" is lit.

With a touch screen field selected (highlighted yellow):

- Turn the knob left/right to lower/raise the value
- Press the knob to toggle between fine and coarse (1, 2 ,5-step increment) adjustments



Zoom

The WavePilot "Zoom" is a submode to some of the other modes that involve the display of traces, such as History Mode or Spectrum Analyzer.

- Press the Zoom button to put WavePilot into Zoom mode.
- Turn the SuperKnob to adjust the scale of the traces.

Table

Likewise, whenever there is tabular display (such as Spectrum Analyzer or Serial Decode), "Table" submode enables you to use the SuperKnob to navigate the table.

- Press the Table button to put WavePilot into Table mode.
- Shift SuperKnob up to Page Up or down to Page Down through the results.
- Turn the SuperKnob to Scroll Up/Down row-by-row.

Tip: In some WavePilot modes, you can shift the SuperKnob left/right to switch between Zoom, Table, and Pan submodes.

Cursors

In "Cursors" mode, you can use the SuperKnob to control the placement of cursor markers.

- Press the Cursors button until the desired cursor type is shown.

  Tip: LEDs around the SuperKnob light to indicate the cursor type: Top, Vertical Absolute; Top+Bottom, Vertical Relative; Left, Horizontal Absolute; Left+Right, Horizontal Relative; None, Off.

- Turn the knob to move the cursor marker(s) to a new position on the trace. Relative cursors will track together.
- To adjust the placement of Relative cursors:
  - Press the SuperKnob to enter "Adjustment" mode.
  - Shift the knob up/down or left/right to select a single marker.
  - Turn the knob to adjust the marker position relative to the other marker.
  - Press the SuperKnob again to return to "Tracking" mode.

Decode

Press the **Decode** button to open or close the Serial Decode dialogs.

Tip: When a decoder is turned on, switching to Table mode will enable you to use the SuperKnob to scroll up/down the result table.

Spectrum

The **Spectrum** button turns on/off the Spectrum Analyzer.

Shifting the Superknob left/right will switch the options to Spectrum Zoom mode, Pan mode, or Table mode.

- In Spectrum "Pan" mode, push the SuperKnob to center the FFT trace around the frequency selected in the Spectrum flyout menu.
- In Spectrum "Table" mode, shift the SuperKnob up/down to Page Up or Page Down the table of Peaks and Markers.
- In Spectrum "Zoom" mode, turn the SuperKnob to adjust the scale of the Spectrum trace.

WaveScan

"WaveScan" mode enables you to use the SuperKnob to control the WaveScan® Search & Find display.

- Press the WaveScan button to switch on/off WaveScan mode.
- Turn the SuperKnob left/right to "scroll" through the WaveScan trace.

LabNotebook

In "LabNotebook" mode, the SuperKnob can be used to select and review Notebook Entries.

- Press the LabNotebook button to open/close the LabNotebook dialog.
- Press the LabNotebook button again to display the first Notebook Entry in the list. Subsequent presses will display the following entry, and so forth.
- Turn the SuperKnob to scroll the list and select a particular entry, then press the knob to recall the oscilloscope settings associated with the entry (Flashback Recall).

History

The History button turns on/off History mode.

On the History flyout menu, a Capacity control is shown. Turn the SuperKnob to adjust the Capacity setting.



Note: A default value for the Capacity control is generated based on your instrument's specific Acquisition Memory Setting (it also varies based on specific setups, configurations, and settings at any given time).

Rotating and Tilting the Display

Rotate Display

The display panel can be rotated 90° to a portrait orientation, optimizing the display for certain grid styles. The touch screen orientation automatically adjusts to the new position.

To rotate the display up (portrait), slide the release lever at the left side of the display panel, and while holding the lever, rotate the display to the right until it "locks" into position.



To rotate down (landscape), slide and hold the release lever (now at the top of the display panel), then rotate the display to the left. When you have moved it about 45°, you can let go and the display will slowly return to the locked position.

Tilt Display

The display can also be tilted forward or back while in either portrait or landscape position for easier rack or benchtop viewing.

To tilt the angle of the display, firmly grab both sides of the panel. Pull the top of the panel toward you to tilt the display down; Push the top of the panel away from you to tilt the display up.

Signal Interfaces

The instrument offers a variety of interfaces for using probes or other devices to input analog or digital signals. See the product page at teledynelecroy.com for a list of compatible devices.

Analog Inputs

A series of connectors arranged on the front of the instrument are used to input analog signals on channels 1-4.

The channel interfaces power probes and completely integrate the probe with the 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 instrument) gain settings are automatically calculated and displayed based on the probe attenuation.

Probes

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

Passive Probes

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

Active Probes

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

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

Most active probes match probe to oscilloscope response automatically using probe response data stored in an on-board EEPROM. This ensures the best possible combined probe plus oscilloscope channel frequency response without the need to perform any de-embedding procedure.

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

LBUS Interface

The LBUS (LeCroy Peripheral Bus) interface provides precise timing synchronization between the oscilloscope and external devices.

It can be used with the optional:

- MS-250 or MS-500 for up-to-36 channels of mixed-signal input.
- WR6Zi-ExtRef-IN/OUT adapter for input/output of a timebase reference clock signal.

Mixed Signal Options

The MS-250 and MS-500 Mixed Signal Oscilloscope Options are complete systems that add digital acquisition and triggering capabilities to a Teledyne LeCroy oscilloscope. The MS-250 enables input of up to 18 lines of digital data; the MS-500 comes in two models enabling input of either 18 or 36 lines. Both operate through connection to the instrument's LBUS interface.

See the *MDA-250 Operator's Manual* and *MDA-500 Operator's Manual* for detailed instructions.

Connecting

1. Connect the smaller end of the Bus cable to the instrument's LBUS interface. Be sure the head is turned so that the wedge fits into the groove at the top of the connector. Fasten the thumb screws.
2. Connect the larger, grooved end of the Bus cable to the MS device. Again, be sure the wedge fits into the groove at the top of the connector. Fasten the thumb screws.
3. Connect one end of the USB cable (that is attached to the Bus cable) to any host port on the instrument, and the other to the USB port on the MS device.
4. Connect the digital leadset(s) to the other end of the MS device.
5. Attach the leads to the digital lines on your DUT. You may use micro-grippers, if desired.

Configuring

Once the device is connected, digital acquisition features are added to the instrument's Vertical menu. Here, you can specify your bus configuration and set the logic threshold using industry standard or custom levels. See Digital for set up instructions.

Oscilloscope Set Up

Powering On/Off

Press the **Power button** to turn on the instrument. The X-Stream application 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 with a signal attached.

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

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

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

Software Activation

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

Firmware

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

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. See Options for instructions on activating optional software packages.

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.

LAN

The instrument accepts DHCP network addressing. Connect a cable from Ethernet port on the side panel to a network access device. Go to Utilities > Utilities Setup > Remote to find the IP address.

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

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

USB Peripherals

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

Printer

Zi series oscilloscopes support USB printers compatible with the instrument's Windows OS. Go to File > Print Setup and select Printer to configure printer settings. Select Properties to open the Windows Print dialog.

External Monitor

You may operate the instrument using the built-in touch screen or attach an external monitor for extended desktop operation.

Note: The oscilloscope display utilizes Fujitsu touch-screen drivers. Because of conflicts, external monitors with Fujitsu drivers can not be used to control the system, only as displays.

Connect the monitor cable to on the of the instrument. Go to Display > Display Setup > Open Monitor Control Panel to configure the display. Be sure to configure the instrument as the primary monitor.

To use the Extend Grids feature, configure the second monitor to extend, not duplicate, the oscilloscope display. If the external monitor is touch screen enabled, the MAUI user interface can be controlled through touch on the external monitor.

Remote Control

Go to Utilities > Utilities Setup > Remote to configure remote control. Connect the devices using the cable type required by your selection. TCP/IP over Ethernet is generally supported.

Sample Clock

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

Reference Clock

To input/output a reference clock signal, connect the optional WR6Zi-ExtRef-IN/OUT adapter to the LBUS connector, then connect a BNC cable from the adapter to the other instrument. Go to Timebase > Horizontal Setup > Clock Source to configure the clock.

Auxilliary Output

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

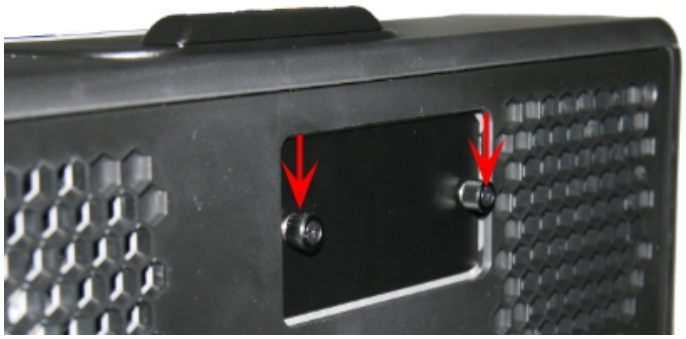
Removable Hard Drive

A removable hard drive option may specified when ordering the oscilloscope (this option requires factory installation). A spare drive is provided in addition to the drive installed inside the instrument.

Caution: Always fully power-down the oscilloscope by unplugging it before removing or re-installing the hard drive. Never install any hard drive other than those provided by Teledyne LeCroy and intended for the instrument.

Removing the Hard Drive

1. Hand-loosen the screws attaching the drive cover and remove the cover.



2. Un-tuck and gently pull the tab to unplug the hard drive from its connection and remove it from the slot.



Re-installing the Hard Drive

1. Place the drive into the slot with the label facing you (as shown above).
2. Firmly press the drive into place to secure its internal connection.
3. Replace the cover so that the inside lip is on the bottom and pushes down on the hard drive.



4. Tighten the screws to secure the cover.

Language Selection

To change the language that appears on the touch screen:

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

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

1. Choose File > Minimize to hide X-Stream 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.
7. Reboot after changing the language.



Note: Other language packs are available from Microsoft's website.

Using MAUI

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

Touch Screen

The touch screen is the principal viewing and control center. The entire display area is active: use your finger or a stylus to touch, drag, swipe, 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
- Grid area
- Descriptor boxes
- Dialogs
- Message Bar

Menu Bar

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

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

If an action can be "undone", the Undo 

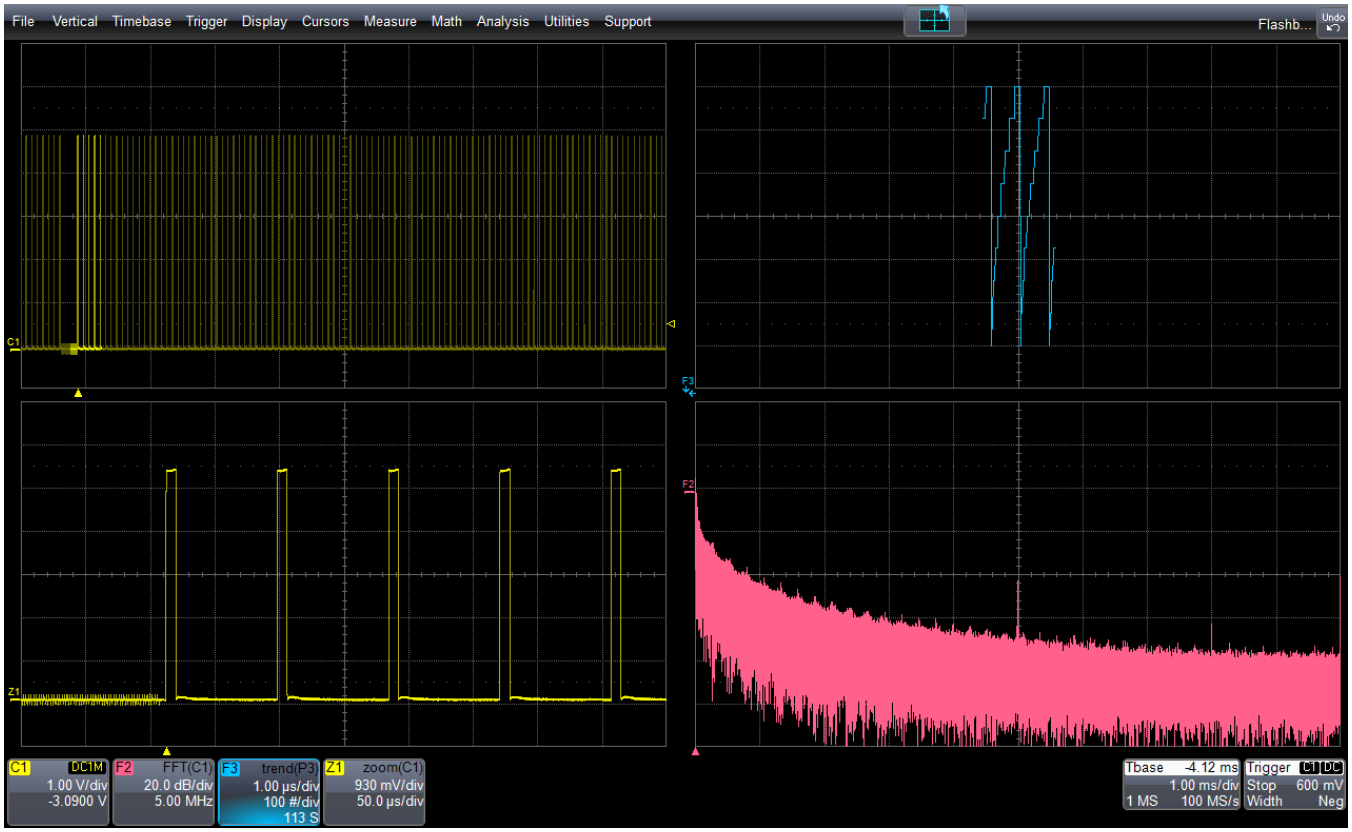
button restores the oscilloscope to the state prior to the action.

Grid Area

The grid area displays the waveform traces. Every grid is 8 Vertical divisions representing 256 Vertical levels and 10 Horizontal divisions each representing acquisition time. The value represented by Vertical and Horizontal divisions depends on the Vertical and Horizontal scale of the traces that appear on the grid.

Multi-Grid Display

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



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

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 Time**, a small triangle along the bottom (horizontal) edge of the grid, shows the time of the trigger. Unless Horizontal Delay is set, this indicator is at the zero (center) point of the grid. Delay time is shown at the top right of the Timebase descriptor box.
	**Pre/Post-trigger Delay**, a small arrow to the bottom left or right of the grid, indicates that a pre- or post-trigger Delay has shifted the Trigger Position indicator to a point in time not displayed on the grid. All Delay values are shown on the Timebase Descriptor Box.
	**Trigger Level** at the right edge of the grid tracks the trigger voltage level. If you change the trigger level when in Stop trigger mode, or in Normal or Single mode without a valid trigger, a hollow triangle of the same color appears at the new trigger level. The trigger level indicator is not shown if the triggering channel is not displayed.
	**Zero Volts Level** is located at the left edge of the grid. One appears for each open trace on the grid, sharing the number and color of the trace.
	Cursor **markers** appear over the grid to indicate specific voltage and time values on the waveform. Drag-and-drop cursor markers to quickly reposition them.

Grid Intensity

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

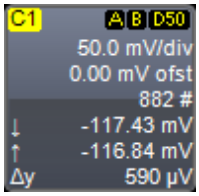
Descriptor Boxes

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

- Inform—descriptors summarize the current trace settings and its activity status.
- Navigate—touch the descriptor box once to activate the trace, a second time to open the trace setup dialog.
- Arrange—drag-and-drop descriptor boxes to move traces among grids.

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

Channel Descriptor Box



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

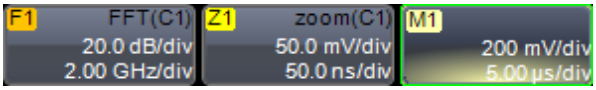
Codes are used to indicate pre-processing that has been applied to the input. The short form is used when several processes are in effect.

Pre-processing Symbols on Descriptor Boxes

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

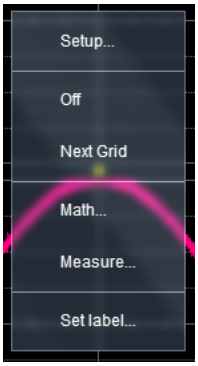
Other Trace Descriptor Boxes

Similar descriptor boxes appear for math (F*x*), zoom (Z*x*), and memory (M*x*) traces. These descriptor boxes show any Horizontal scaling that differs from the signal timebase. Units will be automatically adjusted for the type of trace.



Trace Context Menu

Touch and hold ("right-click") on the trace descriptor box to open the trace context menu, a pop-up menu of actions to apply to the trace such as turn off, move to next grid or label.



Timebase and Trigger Descriptor Boxes

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

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



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

Dialogs

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

Right-Hand Subdialogs

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

Action Toolbar

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



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

Zoom creates a zoom trace of the active trace.

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

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

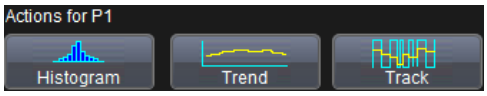
Store loads the active trace into the corresponding memory location (C1, F1 and Z1 to M1; C2, F2 and Z2 to M2, etc.).

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

Next Grid moves the active trace to the next grid. If you have only one grid displayed, a new grid will be created automatically, and the trace moved.

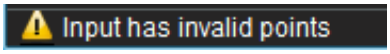
Label opens the Label pop-up to annotate the active trace.

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



Message Bar

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



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



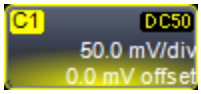
This will be especially evident when you change an acquisition setting that affects the ADC configuration in Normal or Auto trigger mode, such as changing the Vertical Scale, Offset, or Bandwidth. Traces may briefly disappear from the display while the oscilloscope is processing.

Touch Actions

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

Activate

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



Position Cursors

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

Change Trigger

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

Move Trace

To **move a trace to a different grid**, drag-and-drop the trace descriptor box onto the target grid.

Scroll

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

Pan Trace

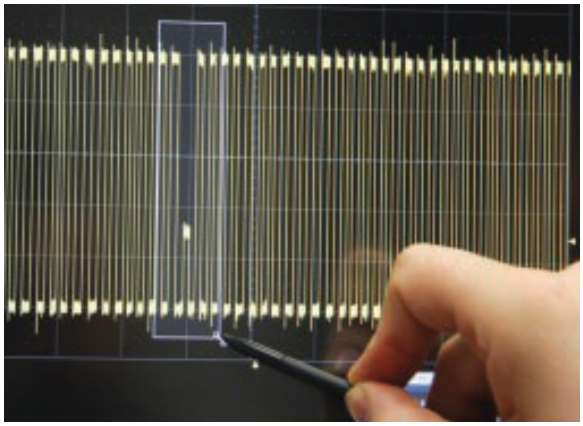
To **pan a trace**, activate it to bring it to the forefront, then drag the waveform trace right/left or up/down. If it is the source of any other trace, that trace will move, as well. For channel traces, the Timebase descriptor box will show the new Horizontal Delay value. For other traces, the zoom factor controls show the new Horizontal Center.

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

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

Zoom

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



See Zooming for other ways to zoom traces.

Working With Traces

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

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

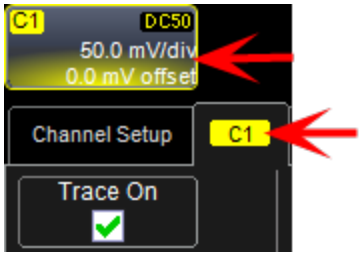
Each visible trace will have a descriptor box summarizing its principal configuration settings.

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



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

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



Active descriptor box matches active dialog tab.

Turning On/Off Traces

Analog Traces

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

From the touch screen, choose **Vertical > Channel *x* Setup** to turn on the trace. To turn it off, touch-and-hold (right-click) on the descriptor box, then from the **context menu** select **Off**. You can also clear the channel checkbox on the Channel Setup or C*x* dialog.

Note: Use the Display menu to change the number and style of grids.

Digital Traces

To turn on the trace from the touch screen, choose **Vertical > Digital*x* Setup** then check **Group** on the Digital*x* dialog.

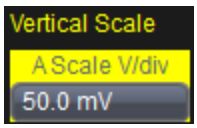
To turn off the trace, clear the Group checkbox.

Other Traces

From the touch screen, touch the **Zoom** or **Math** action toolbar button on the C*x* setup dialog.

Adjusting Traces

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



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

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

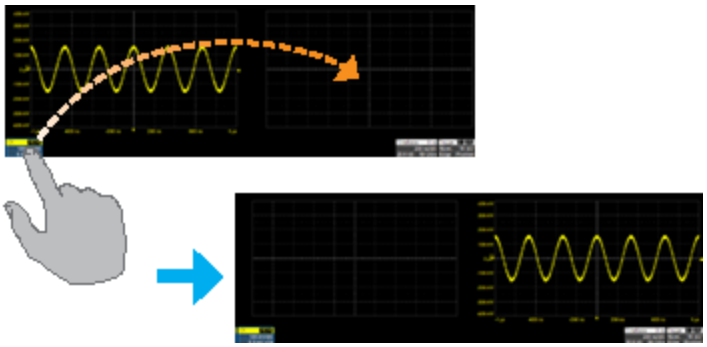
To use the virtual keypad, touch the soft keys exactly as you would a calculator. When you touch OK, the calculated value is entered in the field.

Moving Traces

Use any of these methods to move traces from grid to grid.

Drag-and-Drop

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



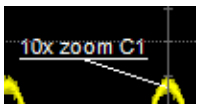
Next Grid Button

Touch twice on the descriptor box of the trace you want moved to open the setup dialog, then touch the Next Grid action toolbar button at the bottom of the dialog. You can also touch and hold (right-click) the trace descriptor box and choose Next Grid from the context menu.

Note: If only one grid is open, a second grid opens automatically when you select Next Grid.

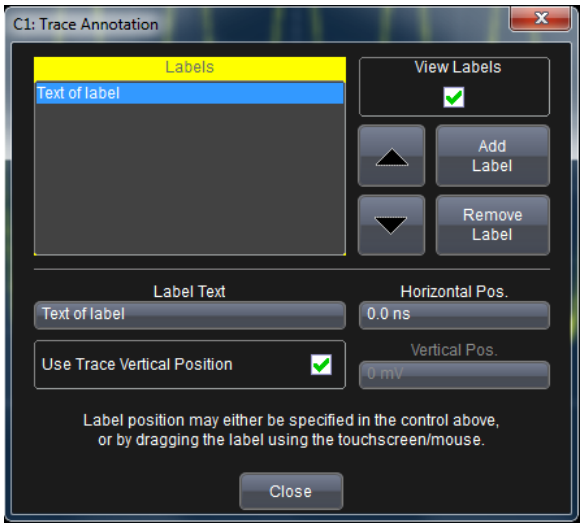
Labeling Traces

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



Create Label

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



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

Reposition Label

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

Edit/Remove Label

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

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

Zooming

Zooms magnify a selected region of a trace by altering the Horizontal Scale relative to the source trace.

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

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

The MultiZoom feature creates time-locked zoom traces for only the waveforms that you choose to include. The zooms are of the same X-axis section of each waveform. As you scroll through a waveform, all included zooms scroll in unison. For more information, refer to MultiZoom.

Zoom Dialog

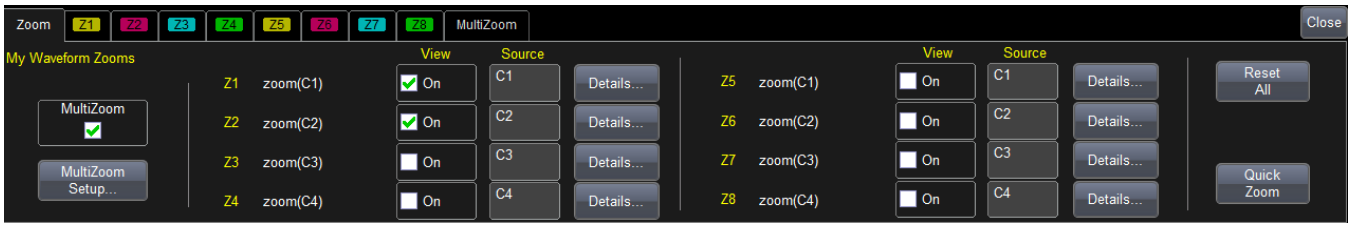
Zoom controls appear throughout the X-Stream software wherever it is likely you may wish to change the display scale of a trace, sometimes as a main dialog, sometimes as a subdialog, such as when setting up:

- Math traces on Fx dialogs
- Memory traces on Mx dialogs

To display the zoom controls, choose **Math > Zoom Setup** from the menu bar, or touch a zoom descriptor box.

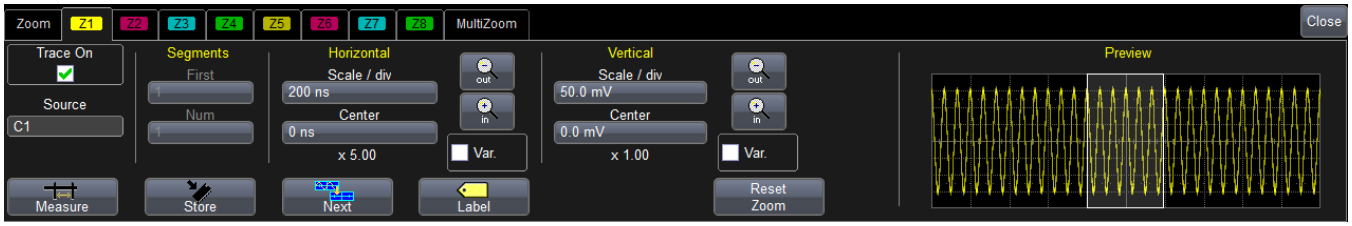
The main Zoom dialog contains selection boxes for turning on/off and changing the source of a zoom. There are also options to:

- **Reset All**, .
- **Quick Zoom**, create a corresponding zoom trace for each open channel trace, same as using the front panel Zoom button.
- MultiZoom



Zx Dialog

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



Trace Controls

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

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

Segment Controls

These controls are used in Sequence Sampling Mode.

Zoom Factor Controls

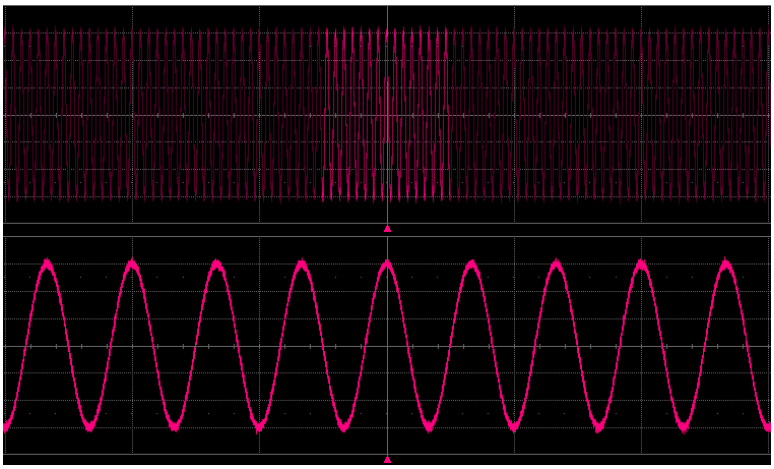
- Out and In buttons increase/decrease zoom magnification and consequently change the Horizontal and Vertical Scale settings. Touch either button until you've achieved the desired level.
- Var.checkbox enables zooming in single increments.
- Horizontal Scale/div sets the time represented by each horizontal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

Creating Zooms

Any type of trace can be "zoomed" by creating a new zoom trace (Z*x*) following the procedures here.

Note: Math and memory traces can be "zoomed" by altering their Horizontal Scale using the zoom factor controls that appear on the F*x* and M*x* dialogs. This method does not create a separate zoom trace.

All zoom traces open in the next empty grid, with the zoomed portion of the source trace highlighted. If there are no more available grids, zooms will open in the same grid as the source trace.



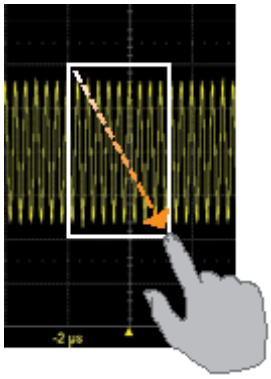
Zoomed area of original trace highlighted. Zoom in new grid below.

Quick Zoom

Use the **front panel Zoom button** to quickly create one zoom trace for each displayed channel trace. Quick zooms are created at the same vertical scale as the source trace and 10:1 horizontal magnification.

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

Manually Create Zoom



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

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

Finally, you can touch-and-hold (right-click) on the descriptor box of the trace you wish to zoom , then choose Math from the context menu. Select the Zoom operator to create a zoom in the next open math function. This method creates a new F*x* trace, rather than a new Z*x* trace, but it can be rescaled in the same manner. It is a way to create more zooms than you have Z*x* slots available on your instrument.

Adjust Zoom Scale

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

Close Zoom

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

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

MultiZoom

MultiZoom creates time-locked zoom traces for only the waveforms that you choose to include. The zooms are of the same X-axis section of each waveform. As you scroll through a waveform, all included zooms scroll in unison.

Set Up MultiZoom

1. Choose **Math > Zoom Setup...** to open the Zoom dialog, then touch the **MultiZoom tab** or **MultiZoom Setup...** button.
2. On the MultiZoom dialog, check **On**.
3. Select all the traces that are **In** the MultiZoom group.



Scroll Waveforms

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

Print/Screen Capture

The front panel Print button captures an image of the display and outputs it according to your Print settings. It can be used to save a LabNotebook, create an image file of waveform traces, or send the display to a networked printer, etc.

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

Print may be used as a screen capture tool by going to File > Print Setup and selecting to print to File, then choosing a graphical format and naming scheme with your Screen Image Preferences. Once configured, just press the Print button or Printer icon, and optionally annotate the image.

You can also use the touch screen to generate a screen capture by choosing File > Save > Screen image and touching Save Now at the right of the dialog. The file is saved using your latest Screen Image Preferences settings.

Acquisition

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

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

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

To avoid aliasing and other waveform distortions, it is advisable (per Nyquist) to acquire at a sample rate at least twice the bandwidth of the input signal. A sample rate 10 times the acquisition bandwidth may be preferable in order to over sample fast signal rise times consistent with the oscilloscope bandwidth.

Those with higher speed acquisition systems may need to manually adjust Bandwidth, Time/div, Sample Rate, and the number of Active Channels to achieve the optimal result. Most Teledyne LeCroy oscilloscopes have a Fixed Sample Rate option, which uses whatever memory is required for the acquisition time, until none remains.

Auto Setup

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

To run Auto Setup:

1. Either press the front panel **Scope Setup button** or choose **Auto Setup** from the Vertical, Timebase, or Trigger menus. All these options perform the same function.
2. From the flyout menu at the right of the touch screen, choose **Auto Setup**.
3. Press the Scope Setup button again or use the touch screen display to confirm Auto Setup.

You can also find a suitable vertical scale for any one channel if it is not obvious how to set the gain and offset to display the channel trace. Follow the same procedure as for Auto Setup, choosing **Cx Find Scale** from the flyout menu.

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



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

Vertical

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

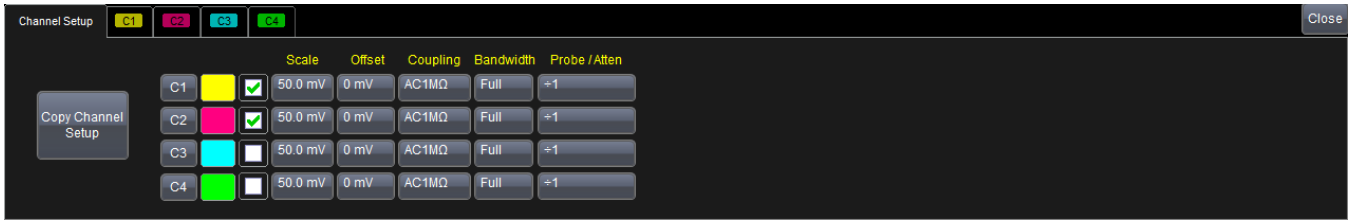
Note: While Digital settings can be accessed through the Vertical menu on instruments with the Mixed Signal option, they are handled quite differently. See Digital.

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

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

Channel Setup Dialog

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



To show/hide the channel trace, select/deselect the checkbox.

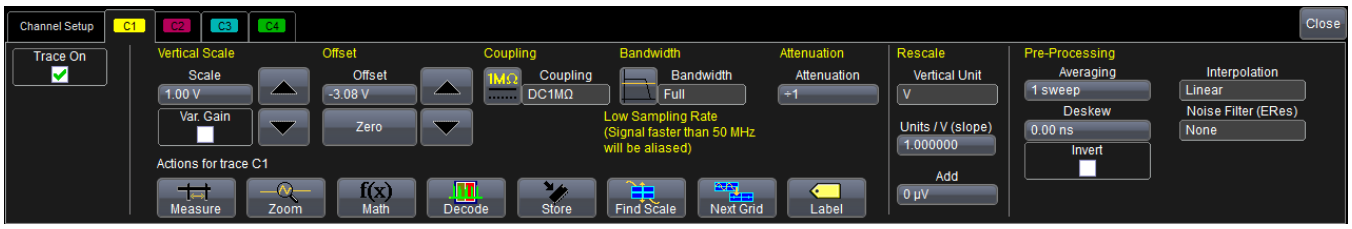
To change the trace color, touch the color block, then choose the new color from the pop-up.

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

To copy settings, touch **Copy Channel Setup** and select the channels to **Copy From** and **Copy To**.

Cx (Channel) Dialog

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



The C*x* dialog contains:

- Vertical settings for scale, offset, coupling, bandwidth, and probe attenuation
- Pre-processing settings for pre-acquisition processes such as noise filtering and interpolation.

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

Vertical Settings

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

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

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

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

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

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

Probe Attenuation and Deskew

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

When a Teledyne LeCroy probe is connected to a channel input, these fields are disabled. For active voltage and current probes, a tab is added to the right of the C*x* tab. The Attenuation field becomes a button to access the Probe dialog. Enter Attenuation on the Probe dialog. Enter Deskew under Pre-Processing settings.

Rescale Settings

The rescale function allows you to apply a multiplication factor, additive constant, and differential vertical unit to the waveform vertical samples.

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

Enter the desired values in Units/V and Add. These two selections provide the same capability as the Rescale math function (y=mx+b) but in a more intuitive, user-friendly format.

Pre-Processing Settings

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

Interpolate applies either Linear or (Sinx)/x interpolation to the waveform. Linear inserts a straight line between sample points and is best used to reconstruct straight-edged signals such as square waves. (Sinx)/x interpolation, on the other hand, is suitable for reconstructing curved or irregular wave shapes, especially when the sample rate is 3 to 5 times the system bandwidth.

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

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

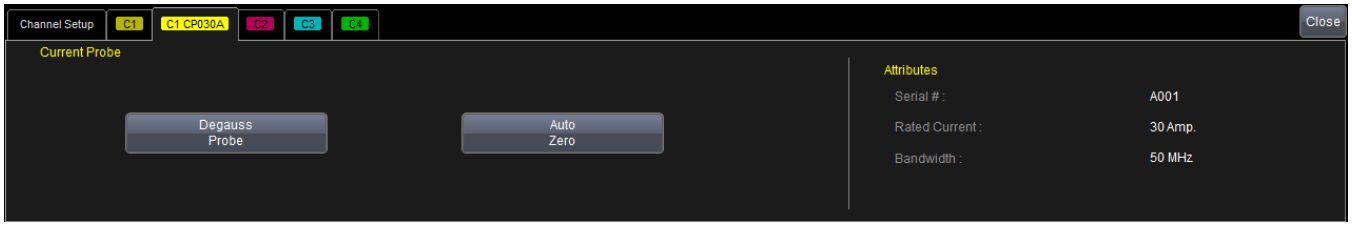
Invert inverts the trace.

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

Probe Dialog

The Probe Dialog immediately to the right of the Cx dialog displays the probe attributes and (depending on the probe type) allows you to AutoZero, DeGauss and make tip selection for probes from the touch screen. Other settings may appear, as well, depending on the probe model.

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



Auto Zero Probe

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

DeGauss Probe

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

Tip Select

If using one of the modular WaveLink probes, specify the type of tip/lead you're using by touching the Tip Select control and making the appropriate selection.

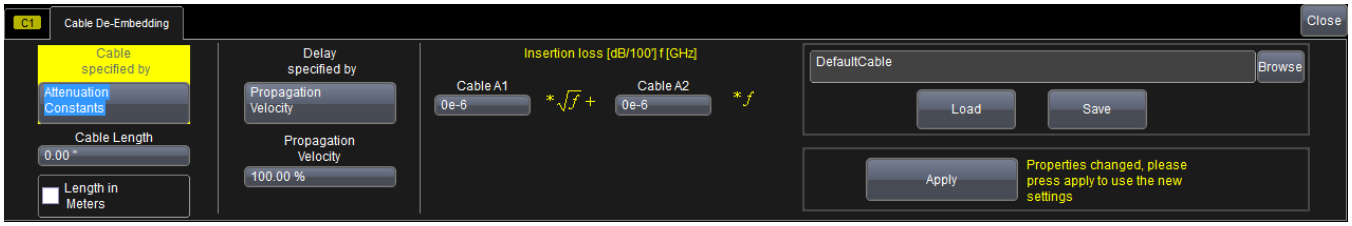
Note: It's crucial to make the tip selection on this field as it results in the amplifier and tip combination having the response calibrated for at the factory. Failure to do so may result in inaccurate measurements.

Cable De-Embedding Option

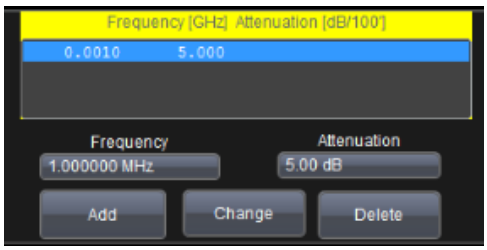
Losses in the cables used in the test setup can reduce the accuracy of your signal (for example, signal amplitude and risetime), as well as introduce Inter-Symbol Interference. These cable effects can dramatically alter measurements and potentially create mask test violations.

Cable De-Embedding, available as an option, allows you to quickly specify the characteristics of the cables (typically found on the cable's data sheet) in your test setup and analyze your signal with the effects of the cables removed. The compensated signal is a more accurate representation of what was actually transmitted.

When Cable De-Embedded is installed and selected from the channel pre-processing settings, the Cable De-Embedding dialog will appear for you to describe the cable being used on the channel.



1. Touch **Vertical > Channel # Setup...** from the menu bar.
2. On the Cx dialog, check **Cable De-Embedding**to display the Cable De-Embedding tab.
3. On the Cable De-Embedding dialog, touch **Cable Specified by** and choose either:
   - **Attenuation Constants.** Enter the cable loss model constants in **Cable A1** and **Cable A2**.
   - **Attenuation Table.** Enter the first set of **Frequency** and **Attenuation** values. Touch **Add** to continue adding rows as needed.



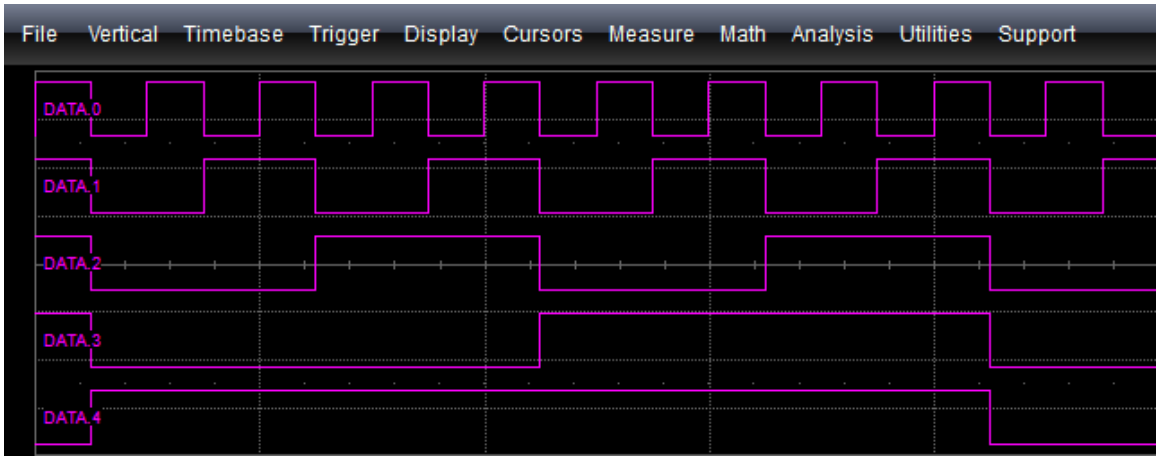
> **Tip:** To edit a row in the table, select the row, change values, then touch **Change**. To delete a row, select the row and touch **Delete**.

4. Touch **Delay Specified by** and choose from Propagation Velocity, Nominal Delay, or Dielectric Constant. Enter the value in the field of the same name.
5. Enter the **Cable Length** in inches.
6. Click **Apply** to initiate the cable de-embedding for the signal on this channel.
7. To save this cable configuration for future use, touch **File Name [DefaultCable]** and enter a name.

Digital (Mixed Signal)

When a Mixed Signal input device is connected to the oscilloscope, digital input setup options are added to the Vertical menu. There are four set up dialogs for each of four possible digital groups, which correspond to buses Digital1 to Digital4. You choose which lines make up each digital group, what they are named, and how they appear on the display.

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

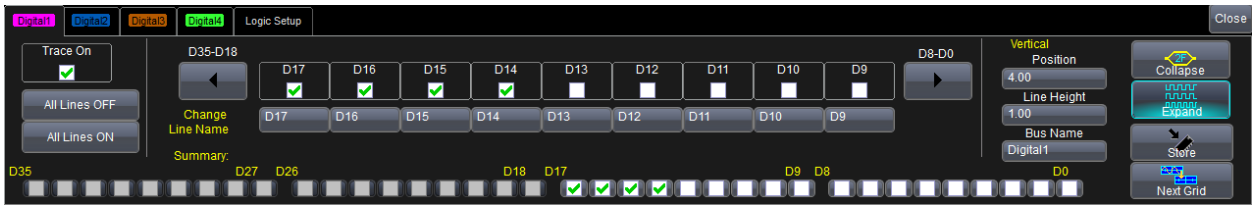


Five digital lines displayed with a Vertical Position +4.0 (top of grid) and a Line Height of 500e-3 division each.

Digitalx (Group) Set Up

To set up a digital input:

1. Connect the Mixed-Signal input device to the test device and the instrument.
2. From the menu bar, choose **Vertical > Digital <#> Setup**.



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

   Alternatively, touch **All Lines ON** to quickly turn on an entire digital bank.

> **Note:** Each group can consist of anywhere from 1 to 18 of the leads from any digital bank. It does not matter if the some or all of the lines have been included in other groups.

4. Check **Trace On** to enable the display.
5. When you're finished on the Digitalx dialog, open **Logic Setup** and choose the **Logic Family** that applies to each digital bank, or set custom **Threshhold** values.



6. Go on to set up the digital display for the group.

Digital Display Set Up

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

1. Set up the digital group.
2. Choose a **Display Mode**:
   - **Expand** (default) shows a time-correlated trace indicating high, low, and transitioning points (relative to the Threshold) for *every* digital line in the group. The size and placement of the lines depend on the number of lines, the Vertical Position and Line Height settings.
   - **Collapse** collapses the lines in a group into their Hex values.
3. In **Vertical Position**, enter the number of divisions (positive or negative) relative to the zero line of the grid where the display begins.The top of the first trace appears at this position.
4. In **Line Height**, enter the total number of grid divisions each line should occupy.

To close digital traces, uncheck the **Trace On** box on the Digital*x* dialog.

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

Renaming Digital Lines

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

1. Set up the digital group.
2. Touch the **Line number field** below the corresponding checkbox. If necessary, use the **Left/Right Arrow buttons** to switch between banks.
3. Use the virtual keyboard to enter the new name, then press **OK**.

   Any active line traces are renamed accordingly.

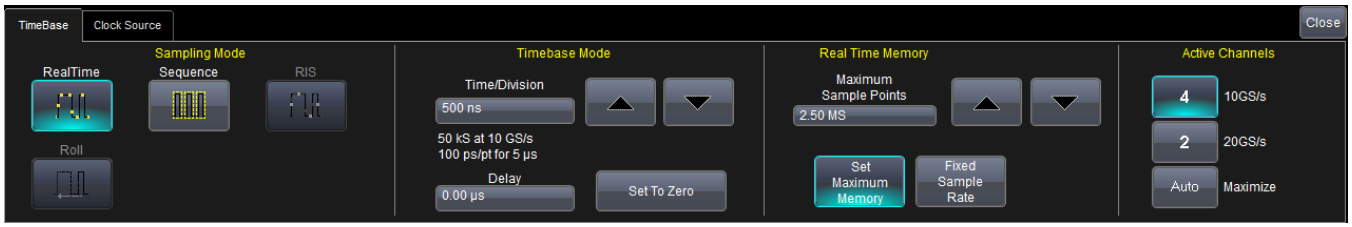
Timebase

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 and clock source 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**.

Timebase Set Up

Use the Timebase dialog to select the sampling mode, memory and number of active channels. You can also use it instead of the Front Panel to modify the Time/Div and horizontal Delay. There are related dialogs for Clock Source.



Sampling Mode

Choose from Real Time, Sequence, RIS, or Roll mode.

Timebase Mode

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

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

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

Set to Zero returns Delay to zero.

Real Time Sampling Rate

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

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

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

Fixed Sampling Rate activates the Sampling Rate field for you to set your own rate. Lowering the rate can extend the acquisition to accommodate slower timebases or longer delays.

Active Channels (Dual-Channel Acquisition)

The Active Channels settings allow you to combine the acquisition capabilities of the leftmost pair of channels (C1 and C2) and the rightmost pair of channels (C3 and C4) to result in two channels with maximum sample rate and memory.

In 4-channel mode, all channels remain active at the default sample rate .

To combine channels, under Active Channels, choose 2 or Auto:

- 2-channel mode turns off waveform acquisition on Channels 1 and 4, although they can still be used for trigger input. Channels 2 and 3 acquire at doubled sample rate and memory .
- In Auto mode, the oscilloscope will allot the maximum memory and sample rate possible based on the activity within each *pair* of channels. As long as only one channel in each of the C1-C2 and C3-C4 pairs is turned on, the maximum rate is used. Turning on both channels in either pair has the same effect as selecting 4 Active Channels.

> Example: In Auto mode, C1 can operate with either C3 or C4 at higher sample rate and memory since they belong to different pairs, and likewise C2. However, C1 cannot operate with C2 without dropping the sample rate, nor can C3 operate with C4.

Refer to the product datasheet for maximum sample rates.

Sampling Modes

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

Real Time Sampling Mode

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

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

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

Roll Sampling Mode

Roll mode displays, in real time, incoming points in single-shot acquisitions that appear to "roll" continuously across the screen from right to left until a trigger event is detected and the acquisition is complete. The parameters or math functions set on each channel are updated every time the roll mode buffer is updated as new data becomes available. This resets statistics on every step of Roll mode that is valid because of new data.

Timebase must be set to 100 ms/div or slower to enable Roll mode selection. Roll mode samples at ≤ 2.5 MS/s. Only Edge trigger is supported.



Note: If processing time is greater than acquisition time, the roll mode buffer is overwritten. The instrument warns, "Channel data is not continuous in ROLL mode!!!" and rolling starts again.

RIS Sampling Mode

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

The maximum effective RIS sampling rate is achieved by making multiple single-shot acquisitions at maximum real-time sample rate. The bins thus acquired are positioned approximately 5 ps (200 GS/s) apart. The process of acquiring these bins and satisfying the time constraint is a random one. The relative time between ADC sampling instants and the event trigger provides the necessary variation.

Because the instrument requires multiple triggers to complete an acquisition, RIS is best used on repetitive waveforms with a stable trigger. The number depends on the sample rate: the higher the sample rate, the more triggers are required. It then interleaves these segments (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.



Note: RIS is not available when the oscilloscope is operating in fixed sample rate Mode.

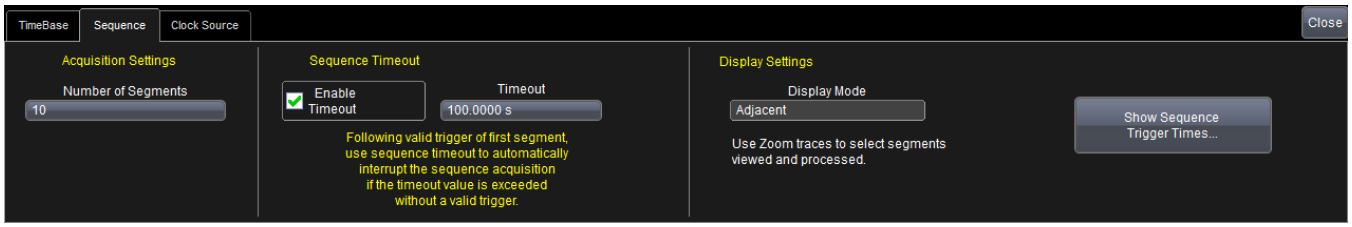
Sequence Sampling Mode

In Sequence Mode sampling, the completed waveform consists of a number of fixed-size segments. The instrument uses the Timebase Sequence settings to determine the capture duration of each segment. The desired number of segments, maximum segment length, and total available memory are used to determine the actual number of samples or segments, and time or points.

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

Sequence Mode Set Up

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



1. From the menu bar, choose **Timebase > Horizontal Setup...**, then **Sequence Sampling Mode**.
2. On the **Sequence tab** under Acquisition Settings, enter the **Number of Segments** to acquire.

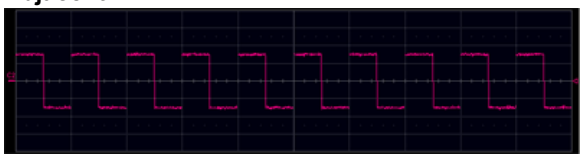
   Note: The number of segments displayed can be less than the total number acquired.

3. To stop acquisition in case no valid trigger event occurs within a certain timeframe, check the **Enable Timeout** box and provide a **Timeout** value.

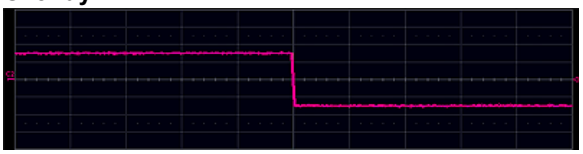
   Note: While optional, Timeout ensures that the acquisition completes in a reasonable amount of time and control is returned to the operator/controller without having to manually stop the acquisition, making it especially useful for remote control applications.

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

Adjacent



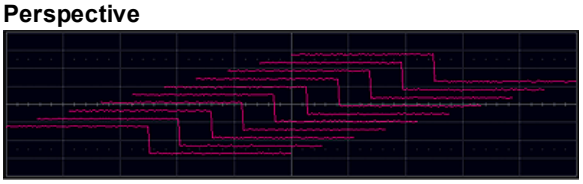
Overlay



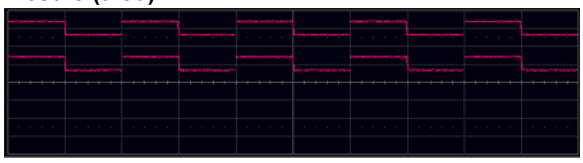
Waterfall (cascaded)



Perspective



Mosaic (tiled)



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

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

View Sequence Segments

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

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

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

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



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

Besides using the Zoom feature, you can also create a Math (F*x*) trace to display individual segments.

1. From the menu bar, choose **Math > Math Setup...**
2. Touch a **Function (Fx) tab** to display its corresponding dialog.
3. On the dialog, touch **Operator1** and select the **Segment** button from the pop-up menu.
4. Touch the **Select** subdialog tab.
5. Touch **First Selected** and choose the first segment to display.
6. Touch **Number of Selected** and enter the total number of segments to display.

To view time stamps for each segment:

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

   Or

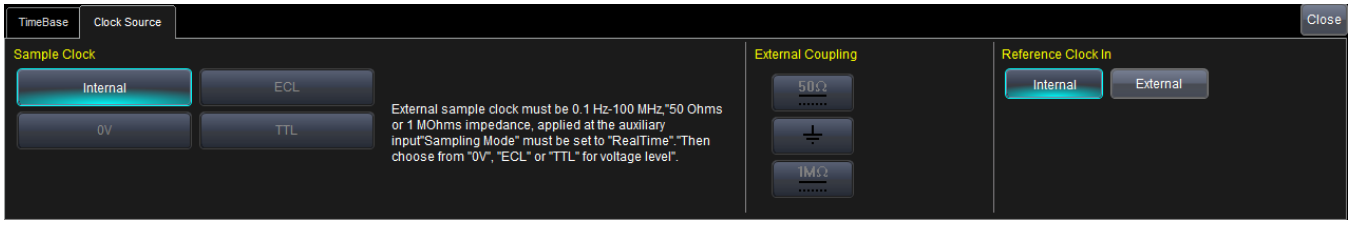
   From the menu bar, choose **Timebase > Acquisition Status**, then open **Trigger time** .
2. Under **Show Status For**, choose **Time**.

Clock Source Settings

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

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

Since the external sampling clock uses the EXT input, an external trigger cannot be used when the external sampling clock is in use.



Clock Frequencies

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

Reference Clock

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

1. Connect a clock source to the optional WR6Zi-Ext-Ref-In/Out adapter. Connect the adapter to the **LBUS** interface.
2. Go to **Timebase > Horizontal Setup** and choose **Real-Time Sampling Mode**.
3. On the **Clock Source tab** under Reference Clock choose **External**.

Sample Clock

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

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

Trigger

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

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

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

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

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

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

Trigger Modes

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

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

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

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

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

Trigger Types

The Trigger Type sets the triggering conditions. Special MultiStage types allow one set of conditions for arming and another (same or different) set for triggering.

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

Basic Types

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

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

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

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

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

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

MultiStage Types

MultiStage triggers establish dependencies between two or more distinct events, some that "qualify" or arm, and others that trigger. The individual events are defined exactly as are the basic types (Edge, Width, Pattern, etc.).

Cascaded establishes a succession of criteria in Stages A-D. The trigger occurs when stages B through D—or whatever is the last defined event—are achieved. In Normal trigger mode, it automatically resets after finding the last defined event. You may apply additional Holdoff by time or number of events between each stage in the cascade.

QualFirst arms the oscilloscope on the A event, then triggers on all subsequent B events, saving each as a Sequence Mode segment.



Note: This trigger is only enabled when using Sequence sampling mode.

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



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

Smart Triggers

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

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

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

Interval triggers upon finding a specific interval, the time (period) between two consecutive edges of the same polarity: positive to positive or negative to negative. Use the interval trigger to capture intervals that fall short of, or exceed, a specified range.

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

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

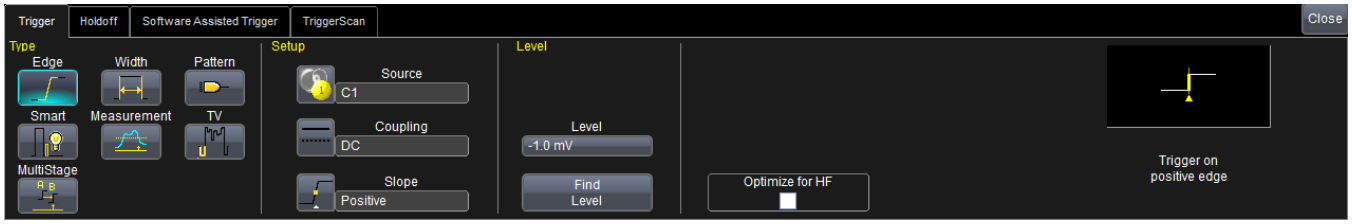
SlewRate triggers when the rising or falling edge of a pulse crosses an upper and a lower level. The pulse edge must cross the thresholds faster or slower than a selected period of time.

Trigger Set Up

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

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

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



Source

For most triggers, the **Source** is the analog channel or digital line to inspect for the trigger conditions.

Pattern triggers may utilize multiple sources (such as a mix of analog and digital signals), and likewise MultiStage triggers may use different sources for the arming and triggering events.

Coupling

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

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

Slope/Polarity

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

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

Level

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

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

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

Hysteresis

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

Conditions (Smart Triggers)

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

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

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

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

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

Patterns

A triggering pattern of low/high states across multiple inputs is used by the Pattern type and MultiStage types that include Patterns as either the arming or triggering event. Patterns may be set on analog channels, digital lines, or a combination or both when Mixed-Signal capabilities are available.

Digital Pattern

The Logic Bus method simplifies pattern set up by utilizing digital groups and logic you have already defined on the Digital Setup dialogs. A digital pattern is set on a single bus (group) manually or by applying a hexadecimal value, while the remaining lines are disabled ("Don't Care").

If you have not set up digital groups, you can set a digital pattern line by line using the Logic method. All available lines remain active for selection.

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



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

Note: As an alternative to a pattern, you may set edge conditions on any line. Touch the **Dx button** and choose the edge. Edge conditions always assume a logical OR in the overall trigger criteria.

As you work, the checkboxes along the bottom of the dialog will change to show the pattern. You can also use these checkboxes to make selections.

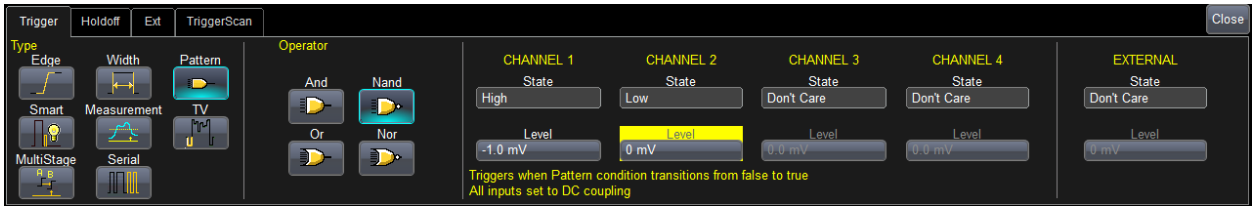
6. If you have not already set a logic threshold, open the **Levels dialog** and select a **Logic Family** for each digital bank from which you've selected lines. To set a custom logic threshold, choose Logic Family User Defined, then enter the **Threshold** voltage and **Hysteresis**.

Note: Digital lines inherit the Logic Setup made when defining digital groups. However, you can change the logic threshold on the Levels dialog. The two settings are linked and always reflect whatever was last selected. Logic thresholds can only be set per lead bank, not individual line.

Analog Pattern

Tip: With the Mixed-Signal option, you can also use the Digital Pattern dialog to set an analog state pattern. Touch Set All To... and select Don't Care. This will eliminate any meaningful digital pattern and activate all the Boolean operators. Use the Left Arrow button to display the C1-EXT group of inputs, and follow the procedure below to set the low/high pattern on each input.

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



2. Select the Boolean **Operator** (AND, NAND, OR, or NOR) that describes the relationship among analog inputs (e.g., C1 must be High AND C2 must be Low).
3. For each input to be included in the trigger pattern, 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.
4. For each input included in the trigger, enter the voltage threshold **Level**.
5. If you've included EXTERNAL as an input, open the **Ext tab** and enter the external trigger **Attenuation**.

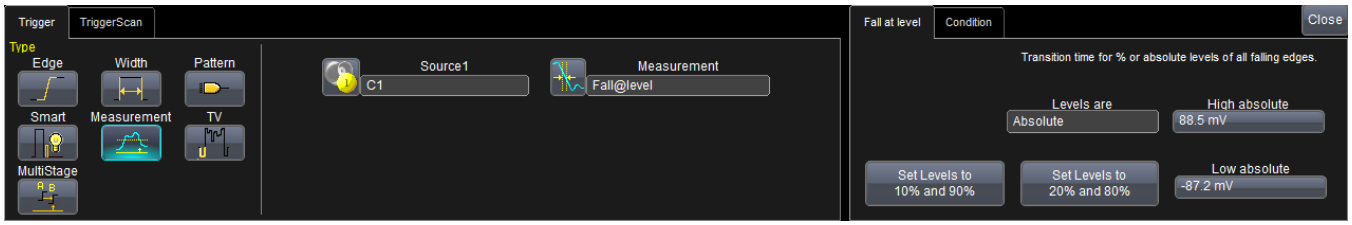
MultiStage Triggers

MultiStage types arm on the A event, then trigger on a subsequent event. The options for the triggering event depend on the type of arming event. See Cascaded, QualFirst and Qualified.

Measurement Trigger

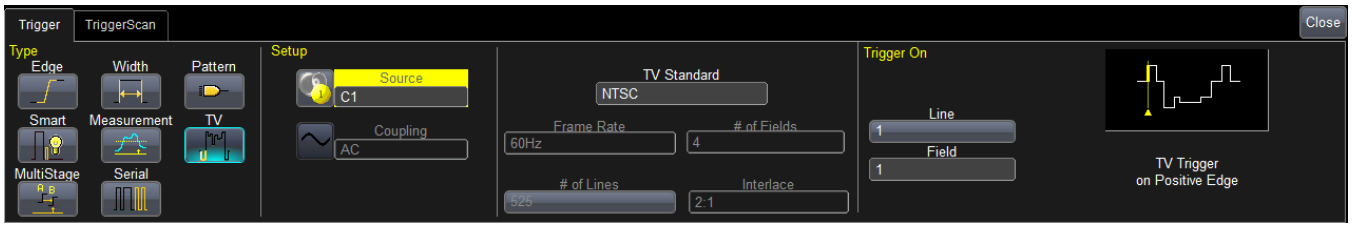
Measurement is not technically a trigger, but permits the instrument to stop and display acquisition based on a post-processing measurement result, similar to a trigger. The available measurements depend on your model instrument and the options installed, and each will offer a different method for specifying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

A measurement can be defined as a standalone trigger condition or as the final condition in a multistage Cascaded trigger.



TV Trigger

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



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

Serial Trigger

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

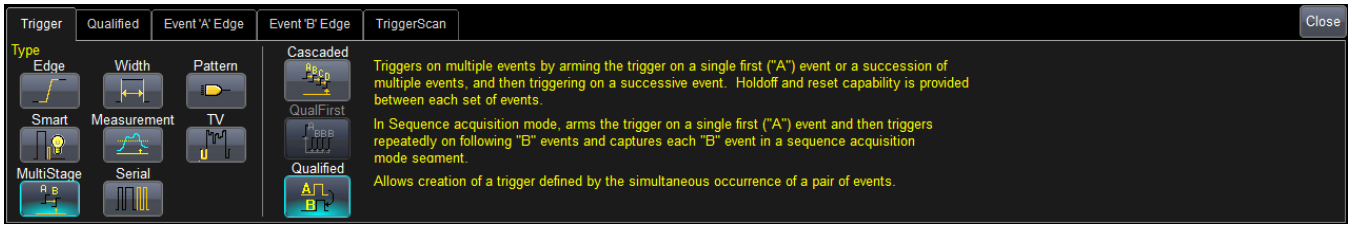
Note: Certain software serial triggers can only be input on certain channels; the set up dialog will instruct you which channel to use. A 1.125 or 3.125 Gb/s Serial Trigger hardware option is available for WaveRunner oscilloscopes (not available on HRO). This option provides capability for 8b/10b symbol triggering. The trigger hardware is factory installed and is permanently connected internally to C4. Complete information on the operation of the hardware serial trigger can be obtained in the *High Speed Serial Triggers Instruction Manual*.

Qualified Trigger

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

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

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



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



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

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

When B is an Edge or Pattern, a time window may be added to the trigger conditions by using the **When B Occurs** buttons:

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

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

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

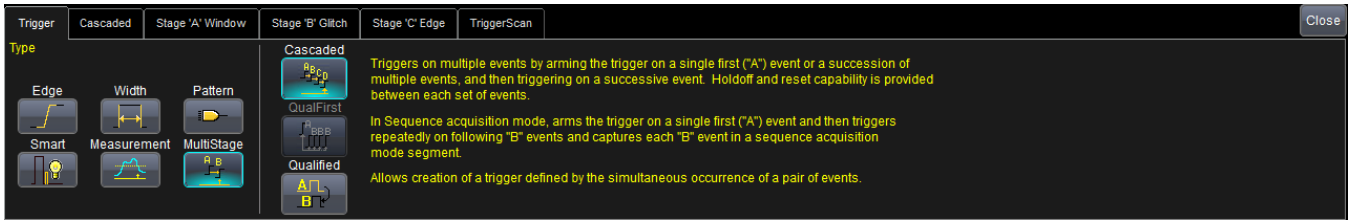
QualFirst Trigger

The QualFirst trigger, which is used in Sequence sampling mode, is set up exactly like the Qualified trigger. The only difference is that the "B" event is captured as a series of sequence segments, rather than a single acquisition.

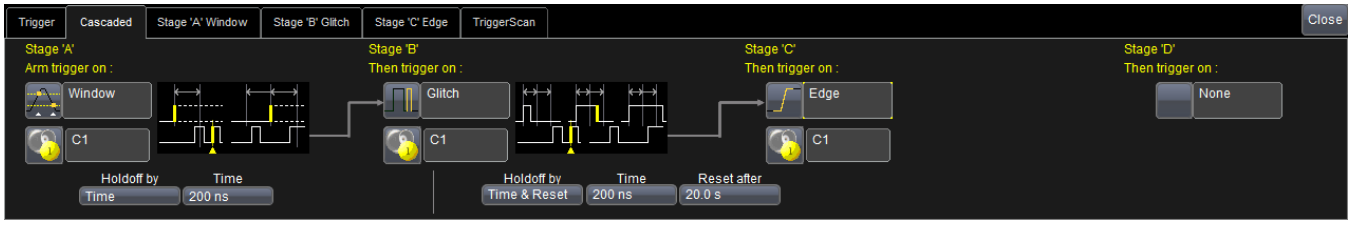
Cascaded Trigger

A Cascaded trigger type establishes a succession of criteria in Stages A-D. The trigger occurs when stages B through D—or whatever is the last defined event—are achieved. In Normal trigger mode, it automatically resets after finding the last defined event. You may apply additional Holdoff by time or number of events between each stage in the cascade.

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



Then, on the **Cascaded dialog** choose the A through D events.



You may apply additional Holdoff at each stage by time or number of events:

- **Time** and **Events** work exactly as they do in other Holdoff counts.
- **Time & Reset** sets a Holdoff time between cascaded events, but resets to the beginning of the cascade if no valid trigger occurs before the reset time expires.

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

Trigger Holdoff

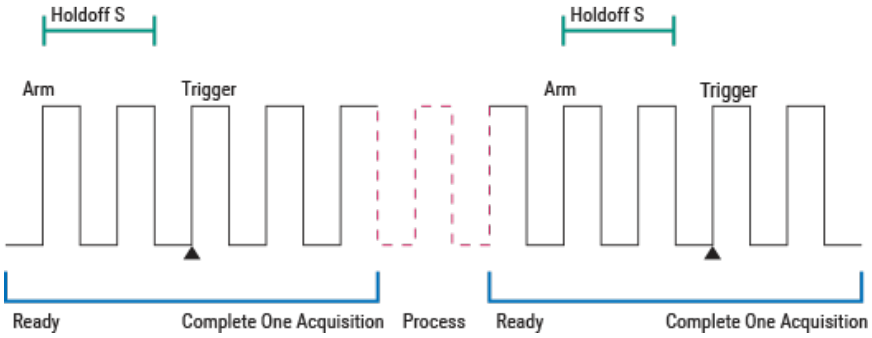
Holdoff is either a period of time or an event count that may be set as an additional condition for Edge and Pattern triggers. Holdoff disables the trigger temporarily, even if the other conditions are met. Use Holdoff to obtain a stable trigger for repetitive, composite waveforms. For example, if the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

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

Hold Off by Time

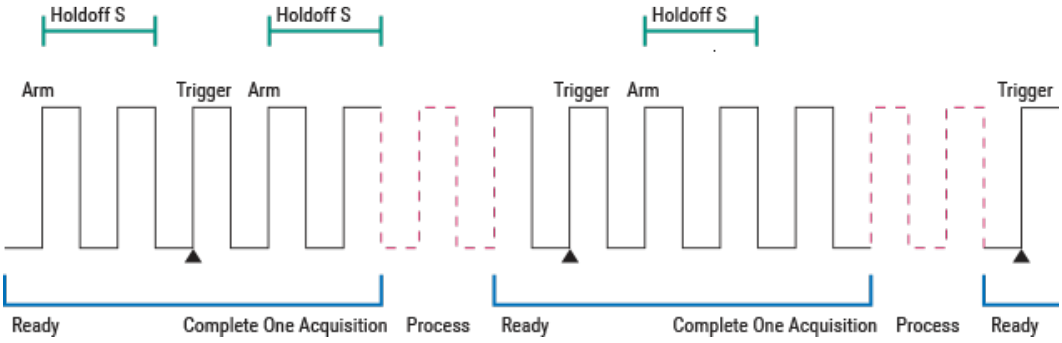
This is a period of time to wait after the arming event before triggering on the next event. The maximum allowed time is 20 seconds; the Holdoff time would otherwise be limited only by the input signal, the coupling, and the instrument's bandwidth.

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



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

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



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

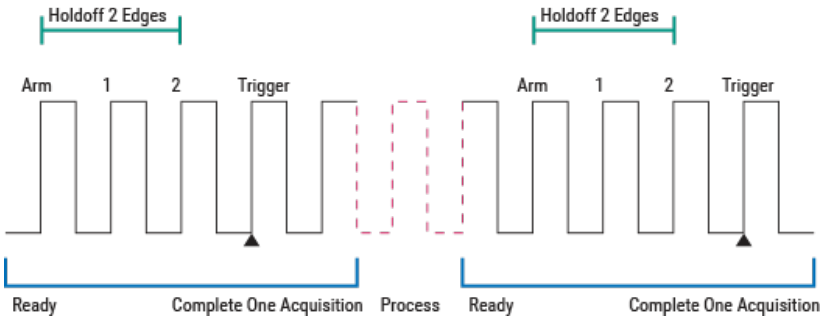
Note: Because there is only one trigger per acquisition, the trigger event will always belongs to the new acquisition. The processing time shown here is for purposes of illustration only.

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

Hold Off by Events

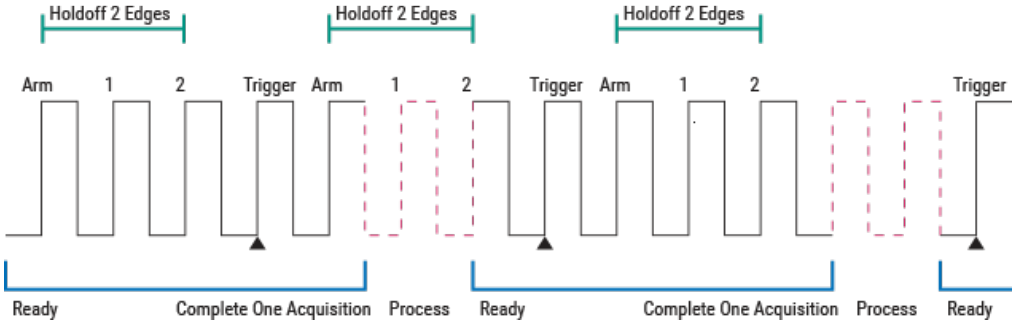
Events refers to the number of times the trigger conditions have been met following the arming event.

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



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

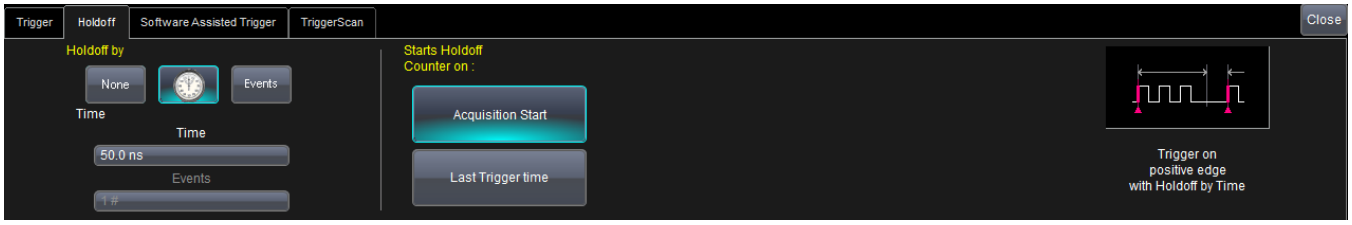
As with Holdoff by time, when a Holdoff by events is counted from the last trigger time, the oscilloscope re-arms immediately following the trigger and begins the Holdoff count. If the count is satisfied by the time the oscilloscope is again ready, the trigger occurs on the next event at the start of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



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

Holdoff Set Up

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



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

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

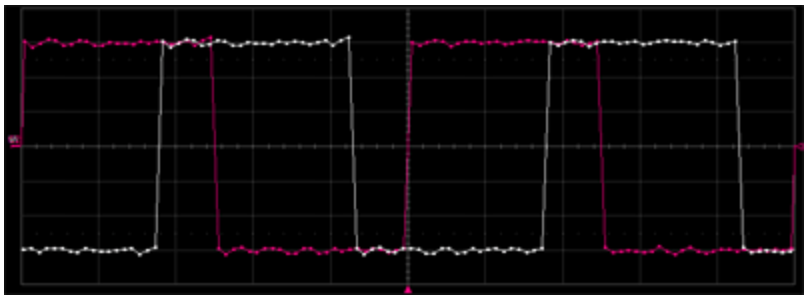
Choose to **Start Holdoff Counter On**:

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

Software Assisted Trigger

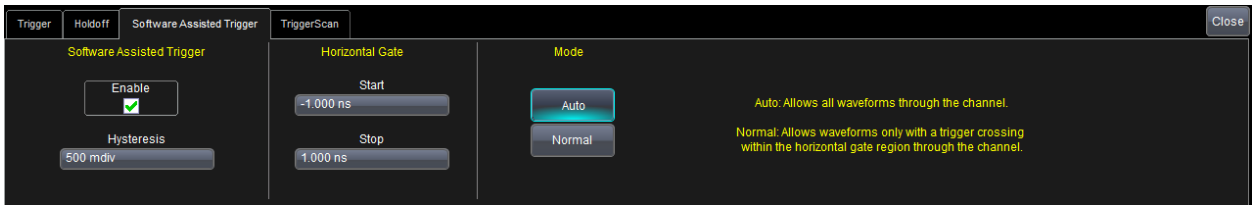
Software Assisted Trigger is used to find the trigger-level crossing point closest to the hardware trigger point. It then adjusts the time offset of the waveform so that it is aligned with the specified trigger level and slope.

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



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

1. From the menu, choose **Triggers > Trigger Setup**, then touch the **Software Assisted Trigger tab**.



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

TriggerScan

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

- **Trains** the system by looking at normal acquired waveforms. During the training, TriggerScan analyzes the waveforms to determine what waveforms normally look like. Using this information, it generates a list of smart triggers to isolate abnormal situations.
- **Loads** the trigger setups from the Trainer and cycles them. As triggers occur, they are overlaid on the screen. All acquisition settings are preserved, and you can use other tools to find the root cause of these anomalies such as WaveScan, Histograms, etc.

Training TriggerScan

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

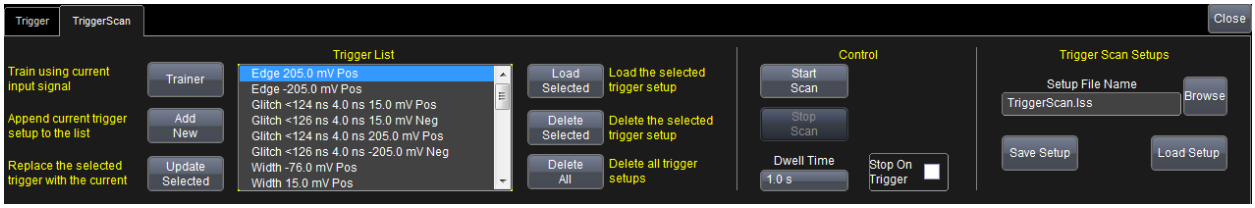
Note: Run the Trainer if you want to change the trigger types or if you change the channel or signal. You must acquire and display at least 3 cycles of a signal before running the Trainer.

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

Modifying Trigger List

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

1. Choose **Trigger > Trigger Setup...** from the menu bar, then open the **TriggerScan** tab.



2. Make any of the following modifications to the Trigger List:
   - Add new trigger: touch the **Trigger** descriptor box and set up the new trigger as desired on the **Trigger** dialog. Then, back on the **TriggerScan** dialog, touch the **Add New** button to append the new trigger to the **Trigger List**.

- Replace a trigger with one manually set up on the Trigger dialog: highlight the setup in the Trigger List and touch the **Update Selected** button.
- Delete a trigger: highlight the setup in the Trigger List and touch the **Delete Selected** button. All trigger setups can be deleted in one step by touching the **Delete All** button.

3. Optionally:
   - Enter a **Dwell Time** the time instrument should wait before loading the next trigger.
   - Check **Stop On Trigger**. You can use this to isolate events as triggers find them.

> **Note:** If you have Persistence enabled, all trigger events are recorded on the display.

Saving TriggerScan Setups

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

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

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

Running TriggerScan

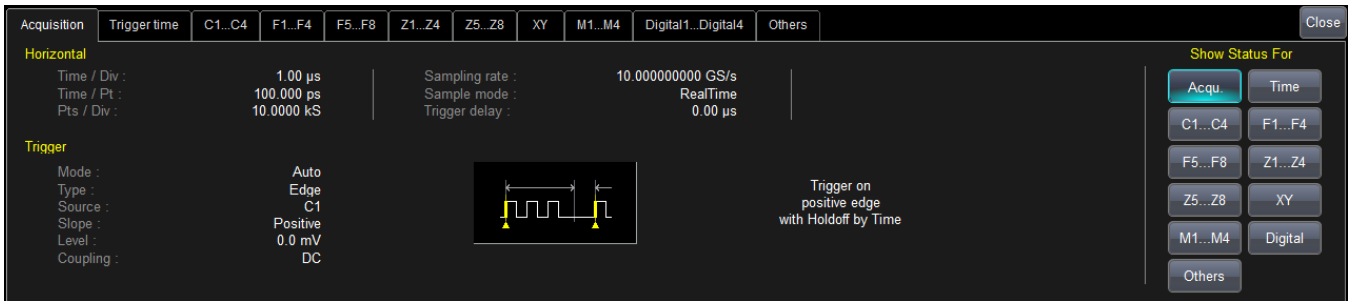
When the Trigger List is set, touch the **Start Scan** button on the Trigger Scan dialog. The instrument automatically cycles through all the triggers in the list.

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

Viewing Status

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

Access the Status dialogs by choosing the Status option from the Vertical, Timebase, Math, or Analysis menus (e.g., Channel Status, Acquisition Status).

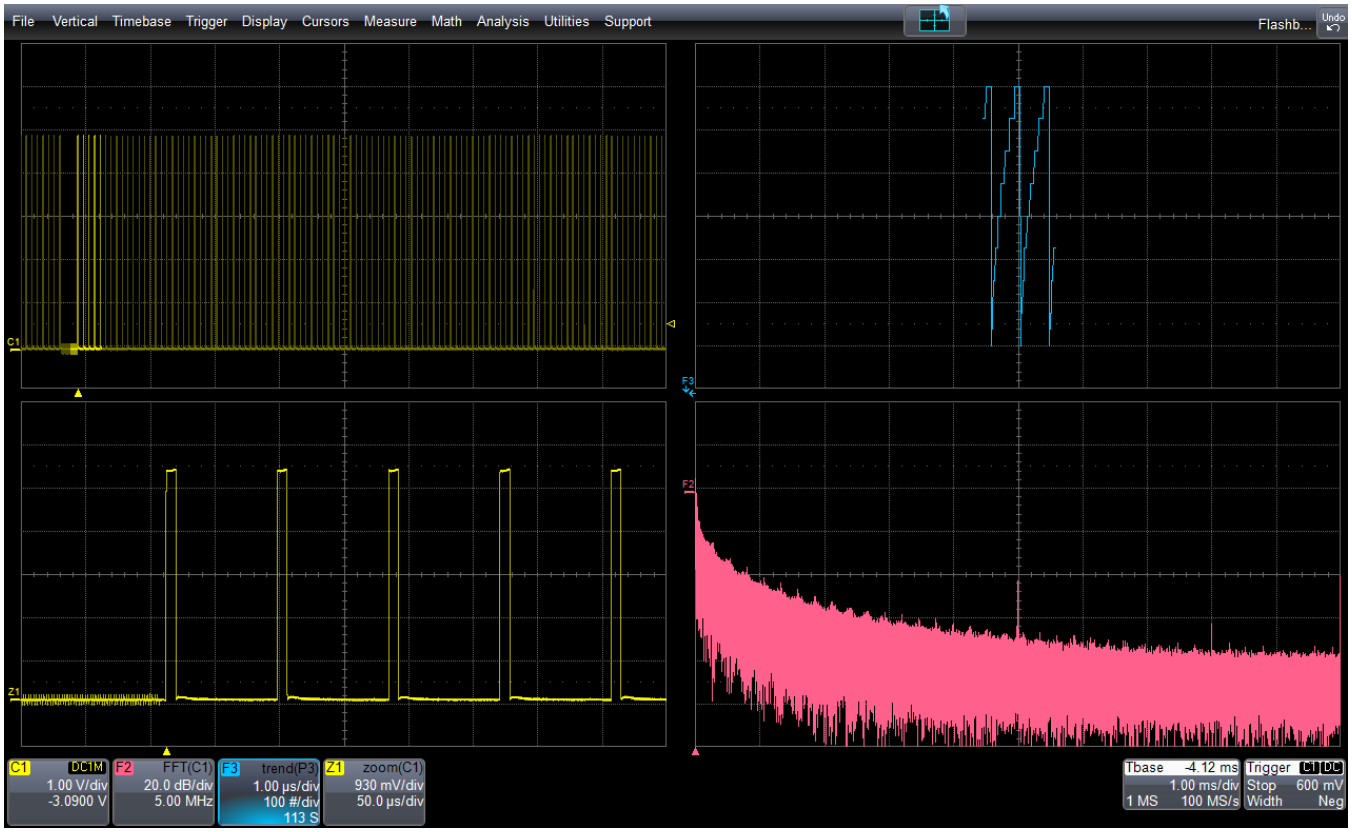


Display

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

Zi series oscilloscopes feature multi-grid display, where each separate grid represents the full number of 256 vertical levels.

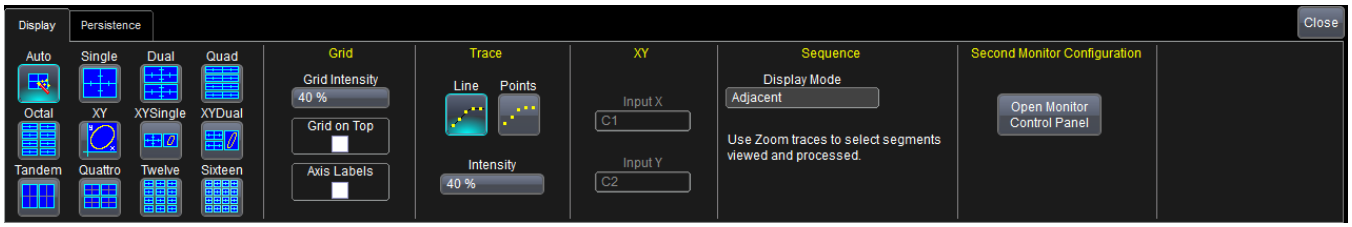
Auto Grid Mode is enabled by default. This feature adds a grid each time a new trace is opened, up to 16 grids, until no more grids are available. There are display options to show all traces on a Single Grid, or to manually divide the display into different numbers and orientations of grids. The Grid Mode icon shows what the result of the selection will be.



Multi-grid display.

Display Set Up

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



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

Grid Mode

Touch the Grid button, then select one of the grid modes. The selection icon shows the number and arrangement of grids.

Grid Mode	Number	Orientation	Notes
Auto (default)	variable	landscape	Automatically adds or deletes grids as traces turned on/off, up to the maximum supported
Single	1	landscape	All traces share one grid
Dual	2	landscape	One top, one bottom
Tandem	2	portrait	One left, one right
Quad	4	landscape	Stacked top to bottom
Quattro	4	landscape	One in each quarter of screen
Octal	8	landscape	Two columns of four stacked top to bottom
Twelve	12	landscape	Three columns of four stacked top to bottom
Sixteen	16	landscape	Four columns of four stacked top to bottom
XY	1	portrait	Single XY type grid
XYSingle	2	portrait	One VT grid left, one XY grid right
XYDual	3	variable	Two VT grids left, one XY grid right

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

Grid Intensity

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

Grid on top superimposes the grid over the waveform.

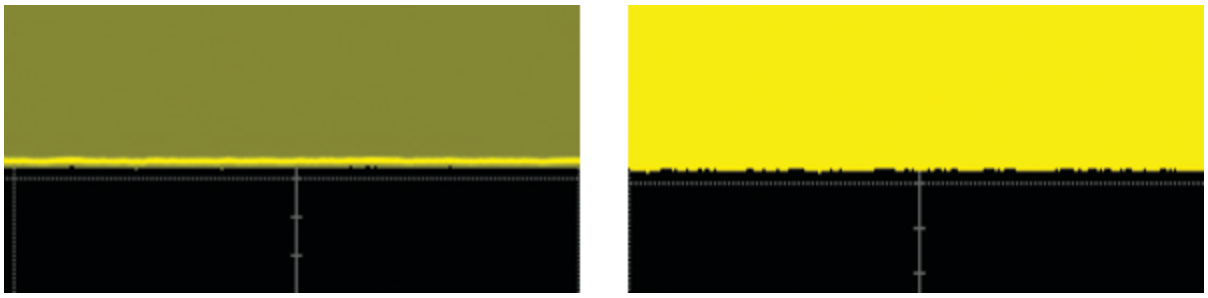
Note: Some waveforms may be hidden from view with the grid on top.

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

Trace Intensity

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

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



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

XY Plots

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

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

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

Sequence Display Mode

Use these settings to select the Display Mode used when sampling in Sequence mode.

Persistence Display

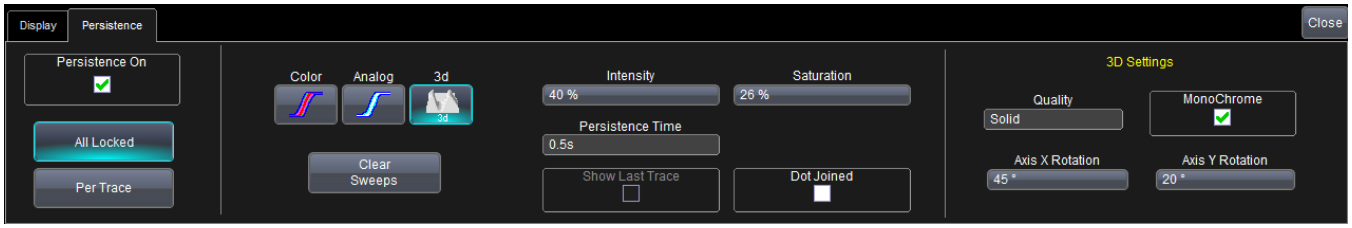
The Persistence feature retains waveform traces on the display for a set amount of time before allowing them to gradually "decay," similar to the analog-style display of old, phosphor screen oscilloscopes.

The display is generated by repeated sampling of events over time and the accumulation of the sampled data into "persistence maps". Statistical integrity is preserved because the duration (decay) is proportional to the persistence population for each amplitude or time combination in the data.

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

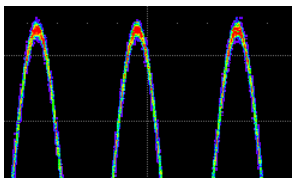
The Ptrace math functions compute a vector trace based on a persistence map that can be used for further analysis. The Phistogram math function can be used to statistically represent the frequency of samples within a vertical and/or horizontal slice of a persistence map.

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

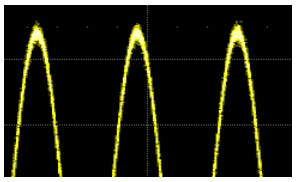


Apply Persistence

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

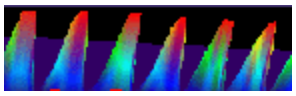


Color Mode persistence works on the same principle as Analog persistence, but instead uses the entire color spectrum to map signal intensity: violet for minimum population, red for maximum population. In this mode, all traces use all colors, which is helpful for comparing amplitudes by seeking like colors among the traces.



In **Analog Mode**, as a persistence data map develops, different intensities of the same color are assigned to the range between a minimum and a maximum population. The maximum population automatically gets the highest intensity, the minimum population gets the lowest intensity, and intermediate populations get intensities in between these extremes. The information in the lower populations (for example, down at the noise level) could be of greater interest to you than the rest. The Analog persistence view highlights the distribution of data so that you can examine it in detail.

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



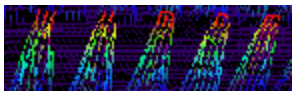
Solid quality can be either color or monochrome. Saturation is set at 50%, with hotter colors indicating highest intensity.



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



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



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

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

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

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

Remove Persistence

To turn off persistence and return to the regular trace style, clear Persistence On or select an individual trace's None (left-most) persistence mode button.

Math and Measure

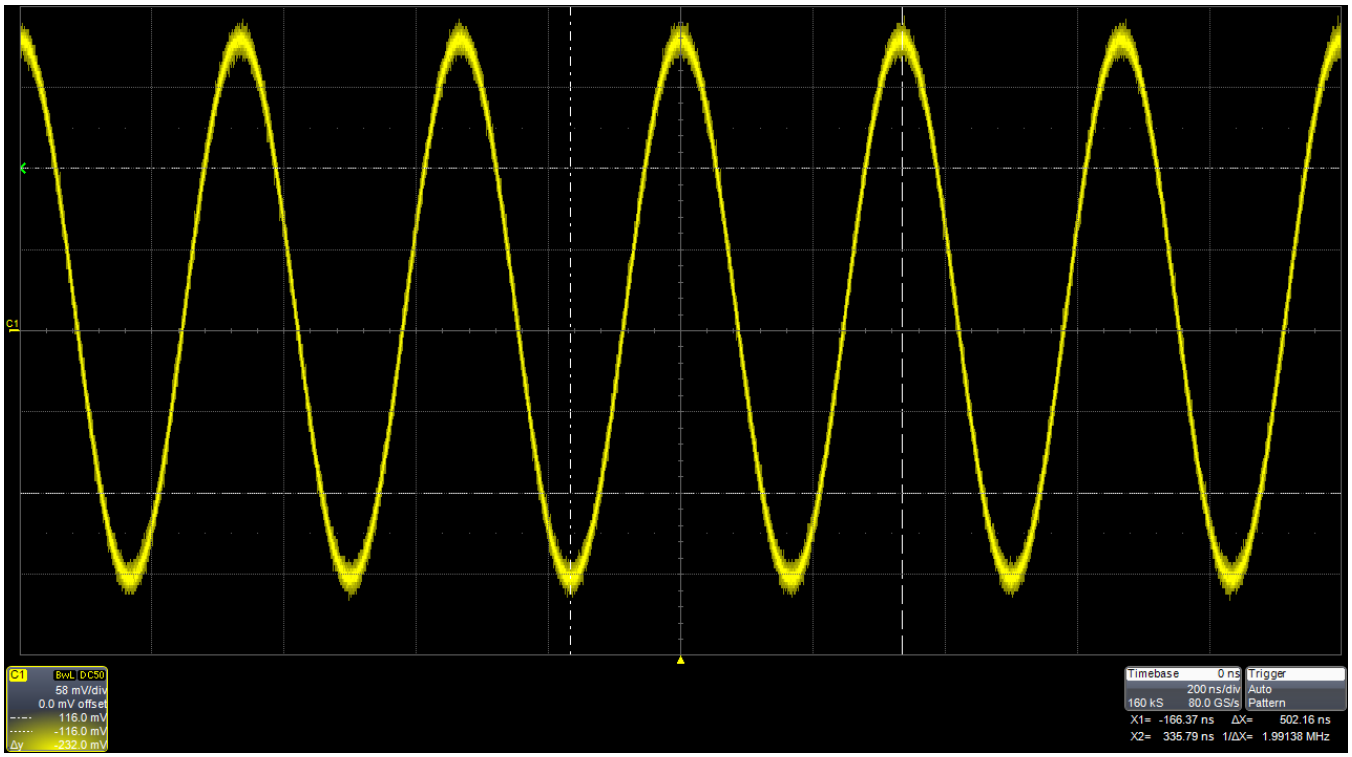
Teledyne LeCroy offers a rich set of standard, pre-programmed tools for the "quickest time to insight" into the characteristics of acquired waveforms. Most instruments calculate measurements for all samples in an acquisition, enabling you to rapidly and thoroughly calculate thousands or millions of parameter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

Cursors

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

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

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



Horizontal Relative cursors.

Cursor Types

Horizontal (Time) cursors intersect points on the horizontal axis.

- Horizontal Abs displays a single, dashed, vertical line. The readout shows the absolute value at the cursor location.
- Horizontal Rel displays two, dashed, vertical lines. The readout may be in absolute values or a delta of the two points, depending on the Show option selected.

Vertical (Amplitude) cursors intersect points on the vertical axis.

- Vertical Abs displays a dashed, horizontal line. The readout shows the absolute value at the cursor location.
- Vertical Rel displays two dashed, horizontal lines. As with Horizontal Relative cursors, the readout may show absolute values or a delta.

The Both Rel(ative) option places both Vertical and Horizontal Relative cursors together.

Apply and Position Cursors

Apply Cursors

To turn on cursors, either:

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

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

To turn off cursors, choose Cursors > Off, or continue cycling the WavePilot Cursor button until they disappear.

Position Cursors

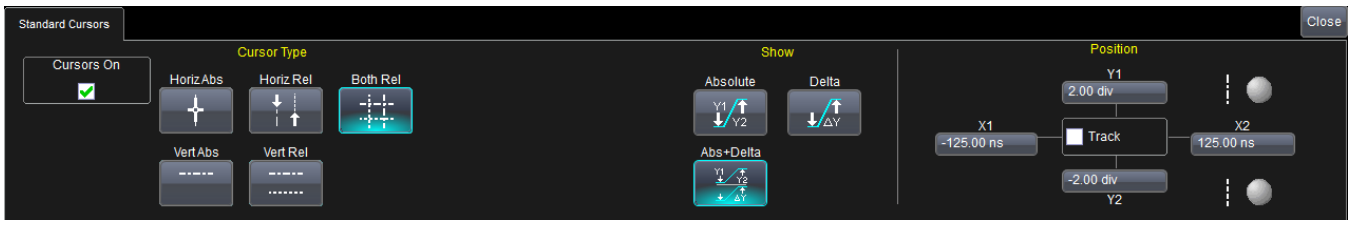
The easiest way to reposition a cursor is to drag-and-drop the cursor marker to a new position. Indicators outside the grid show to which trace the cursor belongs when you have multiple traces on one grid.

Alternatively, turn the front panel Cursor knobs. The top knob adjusts absolute cursors, while the bottom knob adjusts relative cursors. If both vertical and horizontal cursors are turned on, push the knob until the correct line is selected, then turn to move it.

Note: Horizontal Relative cursors are initially placed one on the negative side of the X-Axis (marked by ⇓) and one on the positive side (marked by ⇑). If you move the negative cursor to the right of the positive cursor, or vice versa, the delta is shown in negative increments.

Standard Cursors Dialog

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



Cursors On displays or hide cursor lines. When first checked, the last selected cursor type is displayed.

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

The **Show** controls determine which values are shown in the readout, particularly when using relative cursors:

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

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 and Show settings.

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

XY Cursors Dialog

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

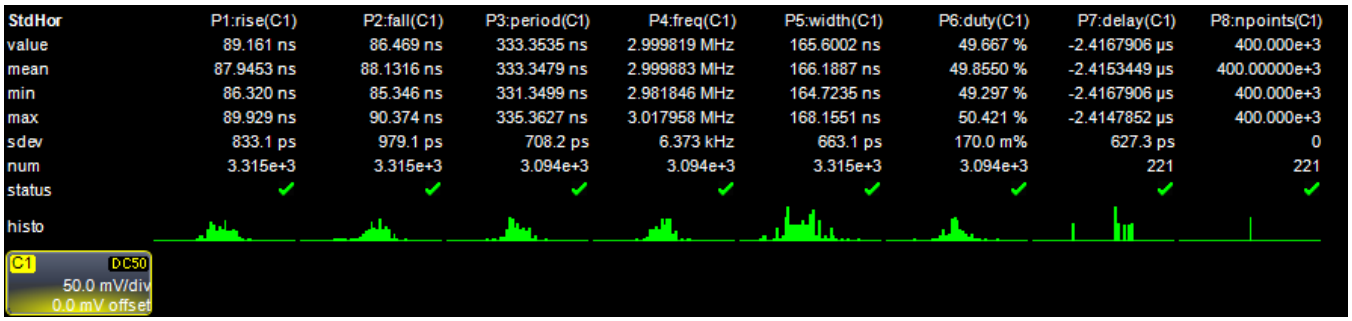
Measure

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

The instrument offers a quick selection of standard horizontal and vertical measurements, or you can create a custom set of parameters (My Measure) drawn from all available measurements.

Measure Table

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



Measurement Status Indicators

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

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

Navigating with the Table

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

Measure Dialog

The Measure Dialog gives quick access to measurement features. Besides turning on/off parameters, use the Measure dialog to display Statistics, Histicons, or Help Markers for standard and user-defined parameter sets. You can also Gate the standard parameter sets to limit the horizontal scope of the measurement. All standard parameters will share the same gate. (Use the P*x* dialogs to gate user-defined parameters.)



Select Measurements

1. To open the Measure dialog, choose **Measure > Measure Setup** from the menu bar.
2. Check **Show Table** to display the measurement readout on screen.
3. Check the selection box for each measurement you wish to display. Touch the **Px button** to define a new measurement.

Standard Parameter Sets

Pre-configured standard parameter sets are available for quick display. Choose either:

- **Std Vertical**: mean, sdev, max., min., ampl, pkpk, top, base.
- **Std Horizontal**: freq, period, width, rise, fall, delay, duty, num points.

To define your own parameter set, choose **My Measure**. Open the Px dialogs to define new parameters. Once parameters are configured, select the **On** checkbox next to each parameter to turn on the measurement.

To quickly begin a new parameter set based on the standard vertical or horizontal sets, touch **Quick Setups** and choose the set to copy. Refine the settings on the Px dialogs.Touch **Clear All** to remove all measurement definitions.

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

Statistics

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

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

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

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

Histicons

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

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

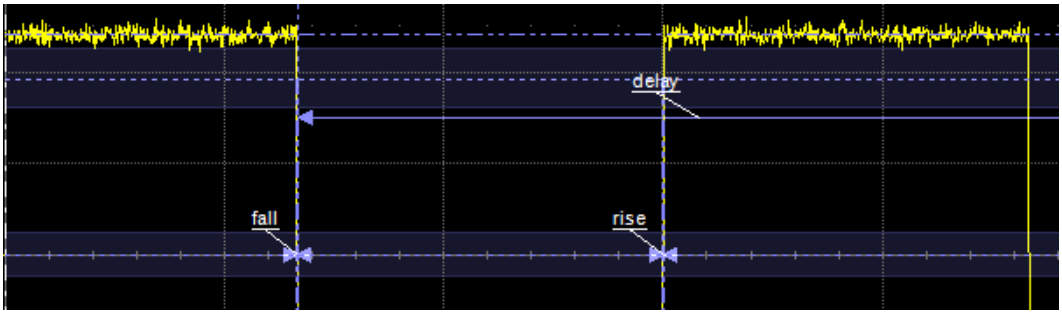
Help Markers

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

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

You also have the option, by means of the Always On checkbox, to leave the markers displayed over traces after you have closed the Measure dialogs or readout table. If you change the set of parameters displayed, the markers will change, as well.

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



Detailed Help Markers on standard horizontal parameters.

Parameter Set Up

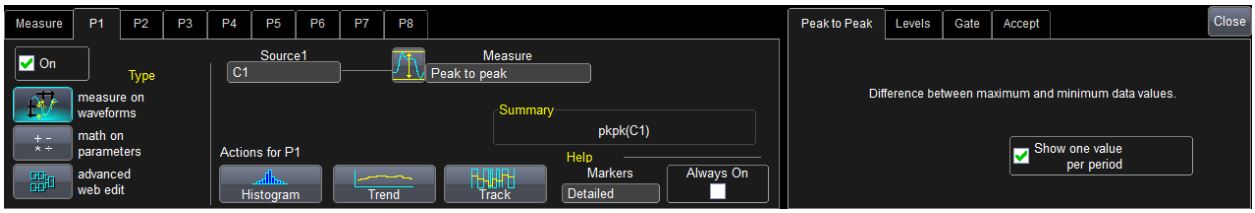
Use the P*x* dialogs to define a new set of measurement parameters ("My Measure").

You may also define parameters using Math on Parameters or the Web Editor.

1. From the menu bar, choose **Measure > Measure Setup**. On the Measure dialog, select **Show Table** to display the readout on screen.

   > **Tip:** To quickly reopen the Measure Setup dialog, touch any cell of the Measure readout table.

2. Choose Measure Mode **My Measure**.
3. Open the dialog **Px** of the parameter you wish to use.
4. Check **On** to enable the parameter and add it to the measurement readout table.



5. Select the Type **Measure On Waveforms**.
6. Touch **Source1** and select the channel, math trace, memory trace, or other waveform to be measured.
7. Touch the **Measure** field and select the measurement from the pop-up menu.
8. Enter the measurement **Level** and **Type** (percent or absolute) when it appears on the parameter subdialogs (e.g., for @Level parameters).
9. For certain time-based measurements, choose to measure on the positive, negative, or both **Slope** (s). For two-input parameters, such as Dtime@level, you can specify the slope for each input, as well as the level and type.
10. Make any further selections on the subdialogs that appear after your Measure selection. These are explained on the dialog and are necessary to fully define the selected measurement.
11. Optionally, turn on Help Markers, or use the parameter subdialogs to:
    - Gate measurements (limit the horizontal scope of the measurement)
    - Qualify measurements (accept only measurements that meet additional conditions)

Gating Measurements

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

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

The quickest way to set a gate is to drag the gate posts from the far left and right of the grid to the desired positions. You can refine this setting down to hundredths of a division by using the **Gate Start** and **Stop** fields on the Gate subdialog.

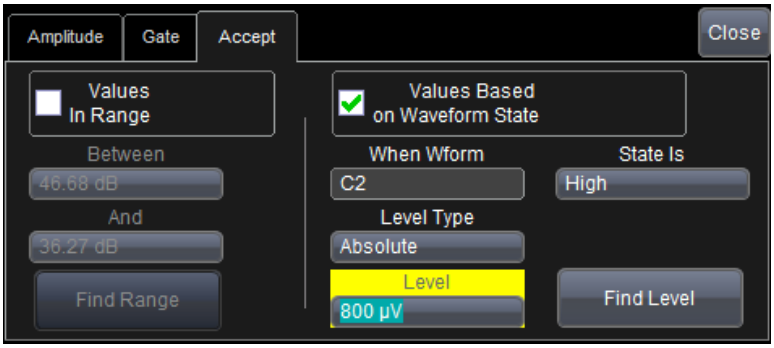
Touch the **Default** button to return gates to the width of the trace.

Qualified Measurements

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

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

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



Range Limited Parameters

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

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

Waveform Gated Parameters

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

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

Using Custom Scripts

On instruments installed with the Advanced Customization option (XDEV), custom scripts may be used to define a measure parameter, math function, or math on parameters. Several scripting "languages" are supported besides VBScript (the instrument's "native tongue"), depending on the application. You can find these options in the Custom submenu of the Measure/Math Selector. Scripts may be imported or written in the instrument's Script Editor window and saved for future use. See Using P Script for a description of the process.

Math on Parameters

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

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

Exclusions

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

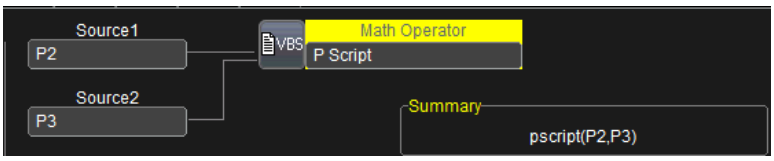
Setting Up Math on Parameters

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

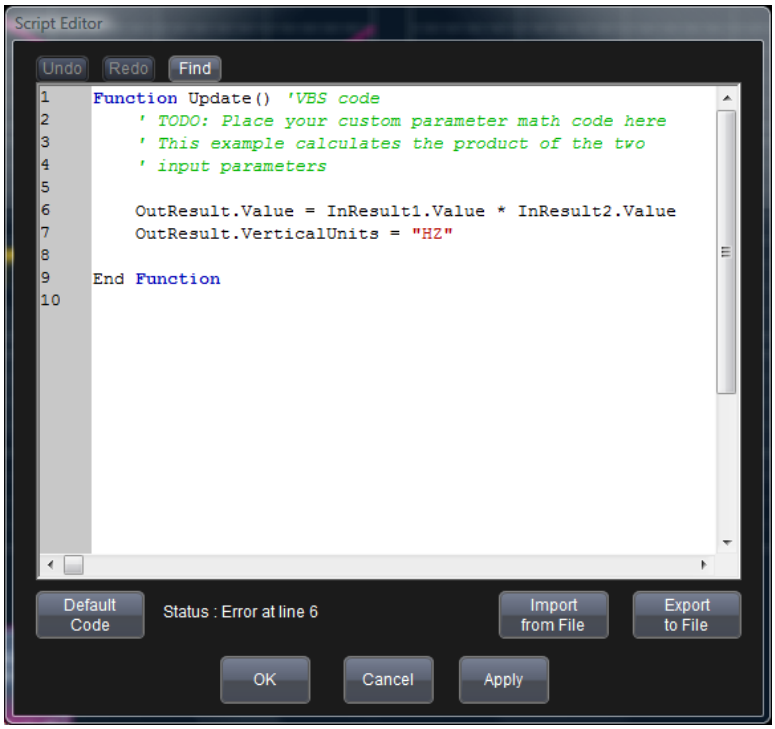
Using P Script (Custom Math on Parameters)

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

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



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



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

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

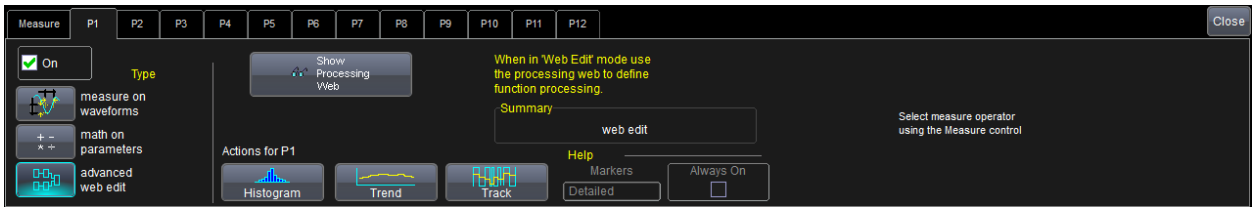
Using Web Editor

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

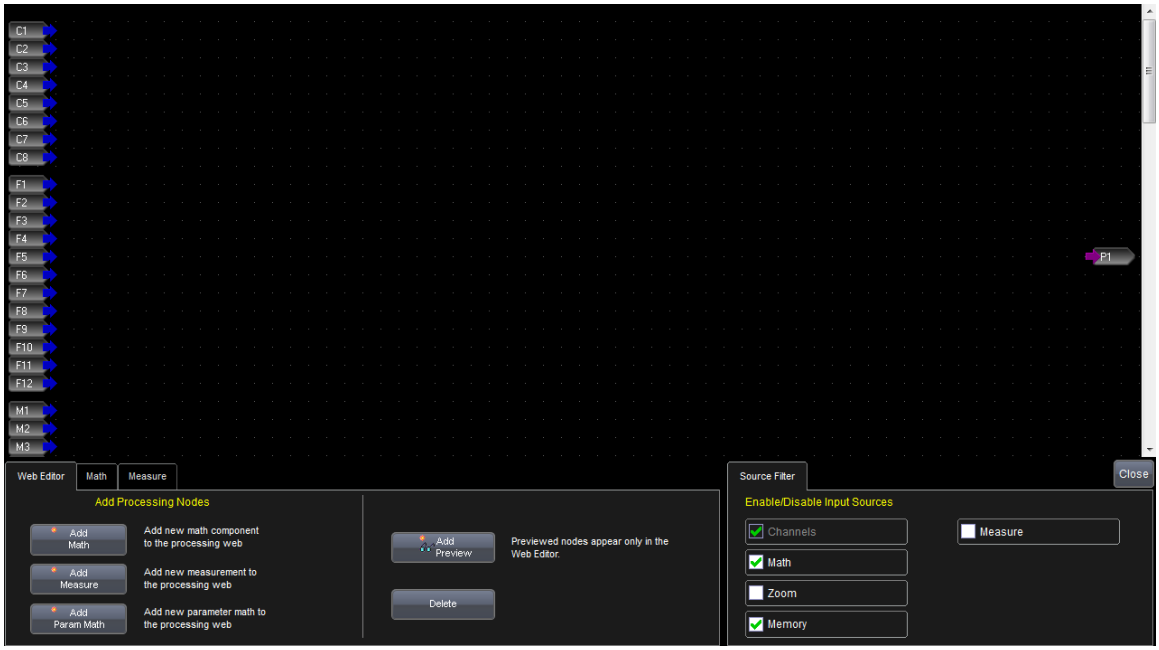
Processes are "programmed" in the X-Stream application simply by dragging and dropping a series of blocks representing different math functions or measurements and connecting them to form a flow chart—a "processing web." The terminal parameters and functions represent the final output of this process.

Create Processing Web

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



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



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

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

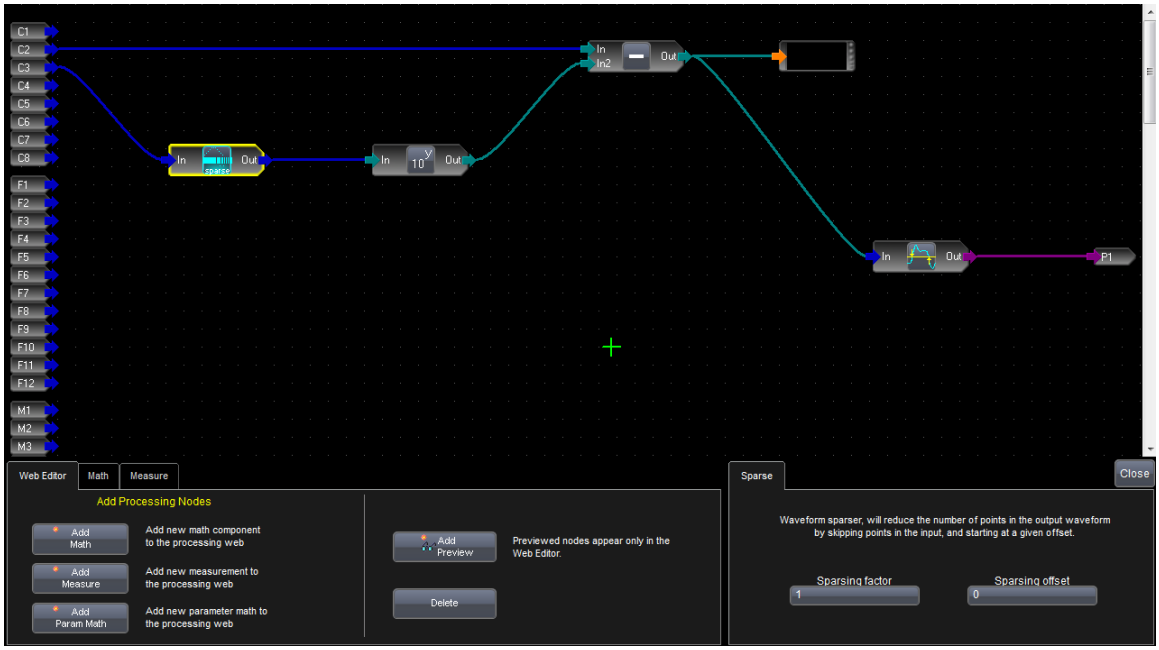
7. From the pop-up, choose the operation or measurement that will occur at that node. A new block appears on the display. Drag the block to a location in the flow.

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

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

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

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



9. If the process requires additional configuration, a subdialog appears next to the Web Editor dialog. Use it to enter the values to apply to that processing node.

10. Continue to add processes as needed, pinning the output of each block to the input of the next block in the flow. The final process should be the same type as the terminal (in this example, a measurement).



Note: The flow does not need to be strictly sequential, as many processes can accept multiple inputs that may or may not have undergone other processes. The only requirement is that the outputs are pinned to acceptable inputs. The color of the input/output arrows indicates which connections are compatible.

11. Drag the output pin from the final process block to the input pin on the terminal.

Preview Outputs

You can add preview "windows" to see the output of a process at any stage in the flow. These previews apply only to the Web Editor.

Select the output pin of the processing block you wish to preview. It will turn bright green. Then, on the Web Editor dialog choose Add Preview.

Delete Process/Connection

To delete any process from the web, select the block then touch Delete. All the connections to/from it are automatically deleted.

To remove a connection between process blocks, touch the line. A scissor icon appears above it. Confirm whether or not to "cut" the connection.

Add Other Terminals to Processing Web

Other measurements or math functions can share the same processing web. For example, you may wish to create a math function of the waveform that would result from a sub-processing node, while the final output of the full process is a measurement parameter.

To add a terminal:

1. Open the Math or Measure dialog behind the Web Editor dialog.
2. Touch the Web Edit icon following the location (P*x* or F*x*). The summary changes to Web Edit, and a new terminal block appears on the Web Editor peg board.
3. Connect the output pin of the desired process to the input pin of the terminal. You can select outputs that are already pinned to other blocks.

Graphing Measurements

Measurements can be viewed in graphical formats to facilitate your analysis:

- Histograms display the distribution of measured values for a given parameter as a bar chart.
- Tracks provide a time-correlated view of a measurement parameter compared to other acquired channels or calculated math traces. A common usage for track is to observe the modulation of a signal, such as amplitude, frequency, or pulse width modulation.
- Trends provide a view of a measurement parameter over an extended period of time and over multiple acquisitions.

Action buttons at the bottom of the the Parameter (Px) dialogs let you quickly draw these plots for the respective measurement.

Although these graphs plot measurement values, they are generated on the display as math functions (F*x*) and can be set up through either the Measure or Math dialogs.

Histogramming

Histograms graphically divide data points into intervals, or bins. These bins are plotted on a bar chart such that the bar height relates to the number of data points within each bin.

Histograms are usually created to visualize the distribution of waveform parameter values over one or more acquisitions. The Phistogram function creates a histogram of values falling within a vertical or horizontal "slice" of a persistence map. Whatever the source, histograms are always created as a math function. The number of sweeps (k#) used to calculate the histogram is shown on the histogram function descriptor box.

Because the shape of histogram distributions is particularly interesting, additional histogram measurements (found in the Statistics sub-menu of the Measure Selector) are available for analyzing these distributions. They are generally centered on one of several peak value bins.

Thumbnail versions of parameter histograms are called Histicons. They are available as a checkbox option on the Measure dialog. Histicons appear on the Measure table, rather than as a new math trace.

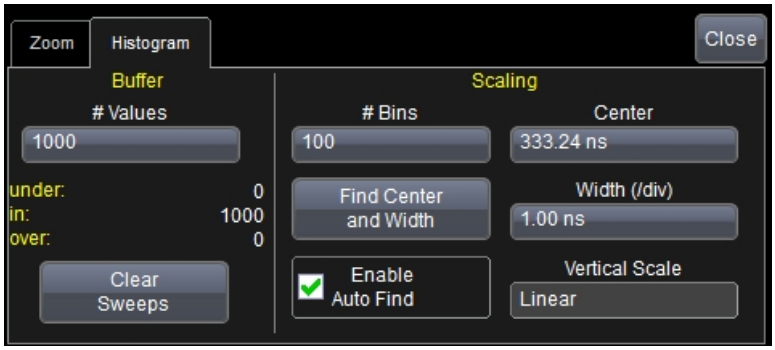
Plot Histogram

Conventional histograms are statistical tools that graph the occurrence of measured events in waveform parameters such as amplitude, frequency, or pulse width. The histogram bar charts are divided into intervals, or bins, where each bin contains a class of similar parameter values.

For a histogram of...	Use...
Different interval widths present in a long data stream	Histogram of Horizontal Parameter Period@level
Cumulative jitter on a long record of a clock signal	Histogram of Jitter Parameter TIE@level
Cycle-to-cycle jitter	Histogram of Horizontal Parameter Dperiod@level

To plot a histogram of a measurement:

1. Open the parameter setup (P*x*) dialog for the measurement you wish to histogram. A quick way to do this is to touch the Measure table cell.
2. Touch the **Histogram** action button at the bottom of the P*x* dialog and choose the function (**Fx**) in which to draw the histogram.
3. Touch the new **hist descriptor box** to display the F*x* dialog, then open the **Histogram subdialog**.



4. Enter the maximum **#Values** in one bin of the histogram. This determines the number of samples that are represented by the bar at full height.
5. Touch **#Bins** and enter the number of bins that comprise the histogram. This determines how many bars appear in the histogram.
6. To let the software determine the range of values represented by each bin/bar, check **Enable Auto Find**, then touch the **Find Center and Width button**. To set your own range, enter **Center** and **Width** values.
7. Choose a **Vertical Scale** method:
   - Linear allows the histogram to build vertically as data accumulates. When the histogram reaches the top of the display, it rescales the vertical axis to keep it on screen
   - LinConstMax keeps the histogram at near full scale and rescales the vertical axis as data is accumulated.

Optimizing Histograms

Histograms are dynamically computed using the last *N* events in the parameter buffer (# Values), up to 2 billion. If the number of bins or values is modified, the histogram is redrawn using the last *N* events required. If the source of the histogrammed parameter is a memory, storing new data to memory effectively acts as an acquisition trigger/sweep and updates the parameter buffer and the histogram.

When the ideal range required to form a good distribution is not apparent, the instrument's range finding function can be used to calculate an optimal range and redraw the histogram using it. The instrument gives a running count of where values fall relative to the range, and if any fall above or below it, the range finder can recalculate the histogram using those values while still in the buffer.

Determining peak value bins, or histogram peaks, is very useful because they indicate dominant values of a signal. However, signal noise and the use of a high number of bins relative to the number of parameter values acquired can yield a spiky histogram, making meaningful peaks hard to distinguish. An optimal bin number is one where the change in bin value is insignificant, and the histogram distribution does not have a jagged appearance. The instrument analyzes histogram data to identify peaks from background noise and histogram definition artifacts such as small gaps, which are due to very narrow bins.

Histogram bins represent a sub-range of waveform parameter values, or events. The events represented by a bin may have a value anywhere within its sub-range. However, *measurements of the histogram itself* assume that all events in a bin have a single value. The instrument uses the center value of each bin's sub-range in all its histogram measurements. The greater the number of bins used to subdivide a histogram's range, the less the potential deviation between actual event values and those values assumed in histogram measurements. Nevertheless, using more bins may require a greater number of waveform parameter measurements (from a greater number of sweeps) to populate the bins sufficiently for the identification of a characteristic histogram distribution.

As with all measurements, the events used to calculate histograms correspond to the portion of the waveform visible on the display, or bounded by the measure gates (the default gates are at the 0 and 10 div positions, enclosing the entire grid, or one full sweep).

In Sequence sampling mode, an acquisition for each segment occurs prior to parameter calculations, and a sweep is equivalent to acquisitions for all segments. If a single segment is selected for viewing, the histogram will be recalculated for only the displayed/gated events in that segment.

For non-segmented waveforms, an acquisition is identical to a sweep. Measurements that yield only one value per acquisition (Amplitude, Area, Base, Cycles, Delay, Delta Delay, Dtime@level, Duration, First, Last, Maximum, Mean, Median, Minimum, Nb Phase, Nb Power, Overshoot+, Overshoot-, Peak-Peak, Phase, Points, RMS, Std Dev., etc.) may not produce the best histograms unless you allow sweeps to accumulate over many acquisitions.

Track and Trend

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

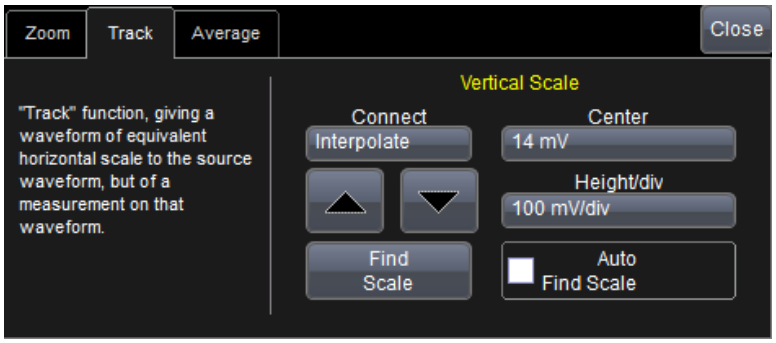
	Track	Trend
Represents	Parameter value vs. time	Parameter value vs. event
Cumulative	Non-cumulative (resets after every acquisition), unlimited number of events	Cumulative over several acquisitions up to one million events
Time correlates to other data	Yes	No
Monitors Frequency domain	Yes	No, points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

Plot Track

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

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

1. Open the parameter setup (P*x*) dialog for the measurement you wish to track. A quick way to do this is to touch the Measure table cell.
2. Touch the **Track** toolbar button at the bottom of the P*x* dialog and choose the function (**F1-Fx**) in which to draw the plot. The track opens in a new grid along with its function descriptor box.
3. Touch the new **Track descriptor box** to display the F*x* dialog, then open the **Track subdialog**.



4. On the Track subdialog, use **Find Scale** to automatically find suitable values, or uncheck **Auto Find Scale** and enter a custom **Center** and **Height/div**.

5. Choose a line **Connect** function of either Interpolate or Extend.

Plot Trend

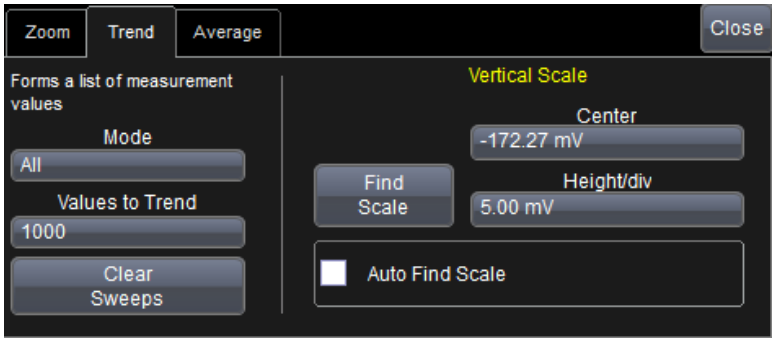
A Trend is a plot composed of a series of parameter measurements in the order the measurements were taken. The vertical units are those of the source parameter, the horizontal unit is measurement number. The Trend contains a single value for each measurement. Think of Trend as a strip chart recorder for your instrument.

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

1. Open the parameter setup (P*x*) dialog for the measurement you wish to histogram. A quick way to do this is to touch the Measure table cell.

2. Touch the **Trend button** at the bottom of the Px dialog and choose the function (**Fx**) in which to draw the plot. The Trend opens in a new grid along with its function descriptor box.

3. Touch the new **Trend descriptor box** to display the F*x* dialog, then open the **Trend subdialog**.



4. Choose a computation **Mode** of All (plots multiple points per acquisition) or Average (plots one point per acquisition). Enter the number of measured **Values to Trend**.

5. To rescale the Trend, use **Find Scale** to automatically find suitable values, or uncheck **Auto Find Scale** and enter custom **Center** and **Height/div** values.

List of Standard Measurements

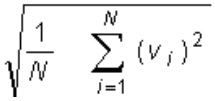
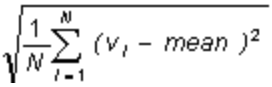
The measurements included standard with your instrument are listed below alphabetically.

Note: Additional measurements may be available depending on the software options installed.

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 > zero contribute positively to the area; values < 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.
Bit Rate	Bit rate of serial data stream.
Cycles (cycles)	Determines number of cycles of a periodic waveform lying between cursors. First cycle begins at first transition after the left cursor. Transition may be positive- or negative-going.
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.
Delta Delay (ddelay)	Computes time between 50% level of two sources.
Dperiod@level (dper@lv)	Adjacent cycle deviation (cycle-to-cycle jitter) of the period measurement for each cycle in a waveform. The reference level for this measurement can be specified.
Dtime@level (dt@lv)	Computes the time between transitions of the selected sources at the specified levels. Only positive going transitions are counted.
Dtrig Time (dtrig)	Time from last trigger to this trigger
Duration (dur)	For single sweep waveforms, dur is 0; for sequence waveforms: time from first to last segment's trigger; for single segments of sequence waveforms: time from previous segment's to current segment's trigger; for waveforms produced by a history function: time from first to last accumulated waveform's trigger.
Duty Cycle	Percent of period for which data are above or below the 50% level of the signal.
Duty@level (duty@lv)	Percent of period for which data are above or below a specified level.
Dwidth@level	Difference (delta) between adjacent widths above or below a specified level.
Edge@level (edge@lv)	Number of positive edges in waveform that cross the specified threshold level.
Edge to Edge	Delta time between some edge on the first source to some (other) edge on a second source.
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.

<table>
<tr><th>Measurement</th><th>Description</th></tr>
<tr><td>Fall time
(fall)</td><td>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.</td></tr>
<tr><td>Fall@level
(fall@lv)</td><td>Fall at level: Duration of pulse waveform's falling edges between user-specified transition levels.
<table><tr><th>Thresh.</th><th>Remote</th><th>Lower Limit</th><th>Upper Limit</th><th>Default</th></tr><tr><td>Lower</td><td>Low</td><td>1%</td><td>45%</td><td>10%</td></tr><tr><td>Upper</td><td>High</td><td>55%</td><td>99%</td><td>90%</td></tr></table>Threshold arguments specify two vertical values on each edge used to compute fall time:
lower = lower thresh. x amp/100 + base
upper = upper thresh. x amp/100 + base</td></tr>
<tr><td>First</td><td>Indicates value of horizontal axis at left cursor.</td></tr>
<tr><td>Frequency
(freq)</td><td>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.</td></tr>
<tr><td>Freq@level
(freq@lv)</td><td>Period of cyclic signal measured as time between every other pair at the specified level. 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.</td></tr>
<tr><td>FWHM</td><td>Measures the width of the largest area histogram peak at half of the populationof the highest peak.</td></tr>
<tr><td>FWxx</td><td>Measures the width of the largest area histogram peak at xx% of the population of the highest peak.</td></tr>
<tr><td>Half Period
(hper)</td><td>Half period of a waveform.</td></tr>
<tr><td>Hist ampl
(hampl)</td><td>Difference in value between the two most populated peaks in a histogram.</td></tr>
<tr><td>Hist base</td><td>Value of the left-most of the two most populated histogram peaks.</td></tr>
<tr><td>Hist max pop</td><td>Peak with maximum population in a histogram.</td></tr>
<tr><td>Hist maximum</td><td>Value of the highest (right-most) populated bin in a histogram.</td></tr>
<tr><td>Hist mean</td><td>Average or mean value of data in a histogram.</td></tr>
<tr><td>Hist median</td><td>Value of the 'x' axis of a histogram that divides the population into two equal halves.</td></tr>
<tr><td>Hist minimum</td><td>Value of the lowest (left-most) populated bin in a histogram.</td></tr>
<tr><td>Hist mode</td><td>Position of the highest histogram peak.</td></tr>
<tr><td>Hist pop@x</td><td>Population at bin for specified horizontal coordinate.</td></tr>
<tr><td>Hist range</td><td>Calculates range (max-min) of a histogram.</td></tr>
<tr><td>Hist rms</td><td>Root mean square of the values in a histogram.</td></tr>
<tr><td>Hist sdev</td><td>Standard deviation of values in a histogram.</td></tr>
<tr><td>Hist top</td><td>Value of the right-most of the two most populated histogram peaks.</td></tr>
<tr><td>Hist X@peak</td><td>Value of the nth highest histogram peak. You supply the value of n.</td></tr>
<tr><td>Hold Time</td><td>Time from the clock edge to the data edge.</td></tr>
<tr><td>Last</td><td>Time from trigger to last (rightmost) cursor.</td></tr>
</table>

Measurement	Description
Level@X (lvl@x)	Gives the vertical value at the specified *x* position. If the *x* position is between two points, it gives the interpolated value. When the Nearest point checkbox is selected, it gives the vertical value of the nearest data point.
MATLAB param	Custom MATLAB function that produces a measurement. Requires XDEV option to edit functions through the oscilloscope GUI using MATLAB Script.
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.
Median	The average of base and top values.
Minimum (min)	Measures the lowest point in a waveform. Unlike base, does not assume waveform has two levels.
N-cycle Jitter	Peak-to-peak jitter between edges spaced *n* UI apart.
NBPhase	Provides measurement of the narrow band phase at a specific frequency of the waveform.
NBPower	Provides measurement of the narrow band power at a specific frequency of the waveform.
None	Disables parameter calculation
Num Points (npoints)	Number of points in the waveform between the measurement gates.
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*.
Peaks	Number of peaks in a histogram.
Percentile (pctl)	Horizontal data value that divides a histogram so the population to the left is xx% of the total.
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.
Period@level (per@lv)	The time between every other pair of at the level specified. 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.

Measurement	Description
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.
Rise@level (rise@lv)	Duration of pulse waveform's rising edges between user-defined transition levels. Threshold arguments specify two vertical values on each edge used to compute rise time: lower = lower thresh. x amp/100 + base upper = upper thresh. x amp/100 + base
RMS	Root Mean Square of data (between gates) calculated using the formula:  Where: vi denotes measured sample values, and N = number of data points within the periods found up to maximum of 100 periods.
Setup	Time from the data edge to the clock edge.
Skew	Time of Clock2 edge (nearest to Clock1) minus time of Clock1 edge. "Clock" refers to specified edge and level of any two sources, data signals included.
Slew Rate (slew)	Slew rate or local dV/dt in a transition zone
Std Dev (sdev)	Standard deviation of the data between the measure gates using the formula:  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
TIE@level (tie@lv)	Difference between the measured times of crossing a given slope and level and the ideal expected time. For Slope you can choose positive, negative, or both. For output units you can choose time or unit interval (UI). A unit interval equals one clock period. The Virtual Clock setup gives you a choice of Standard (1.544 MHz) or Custom reference clocks. You can also use a mathematically derived Golden PLL to filter low frequency jitter. The cutoff frequency is user selectable.
Time@level (time@lv)l	Time from trigger (t=0) to crossing at a specified level.
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.
Total Pop (totp)	Total population of a histogram.
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.
Width@level (wid@lv)	Width measured at a user-specified level.
WidthN (widn)	Time of cyclic signal determined by examining 50% crossings in data input. The widthN is measured from falling edge to rising edge.

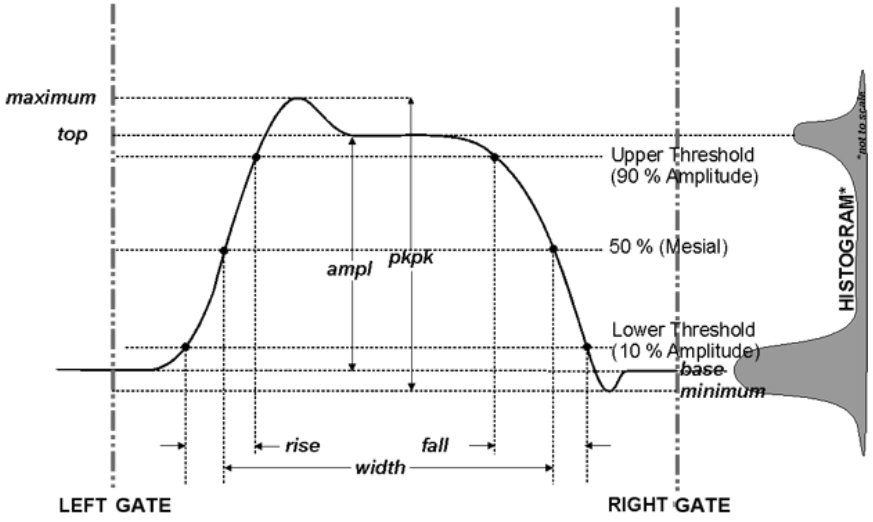
Measurement	Description
X@max	Determines the horizontal axis location of the maximum value between the measure gate.
X@min	Determines the horizontal axis location of the minimum value between the measure gate.

Calculating Measurements

The instrument uses the following methods to calculate measurements.

Determining Top and Base Lines

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



Determining Rise and Fall Times

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

Threshold levels for rise or fall time can also be selected using absolute or relative settings (if @level measurements are included with your model). When absolute settings are chosen, the rise or fall time is measured as the time interval separating the two crossing points on a rising or falling edge. When relative settings are chosen, the vertical interval spanned between the base and top lines is subdivided into a percentile scale (base = 0 %, top = 100 %) to determine the vertical position of the crossing points. The time interval separating the points on the rising or falling edges is then estimated to yield the rise or fall time. These results are averaged over the number of transition edges that occur within the observation window.

Rising Edge Duration

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

Falling Edge Duration

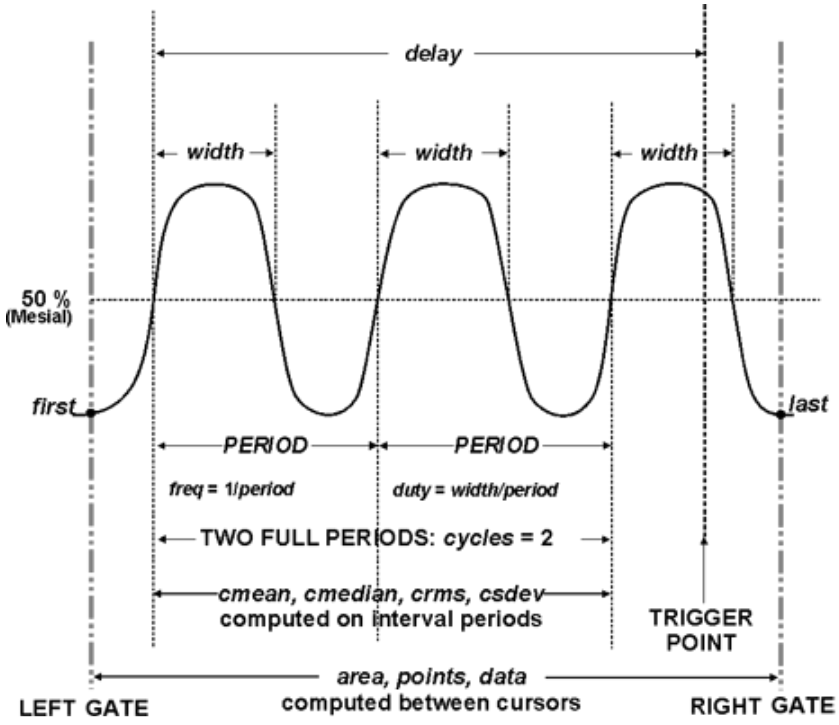
$$\frac{1}{Mf}\sum_{i=1}^{Mf}(Tf_i^{10} - Tf_i^{90})$$

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

Determining Time Parameters

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

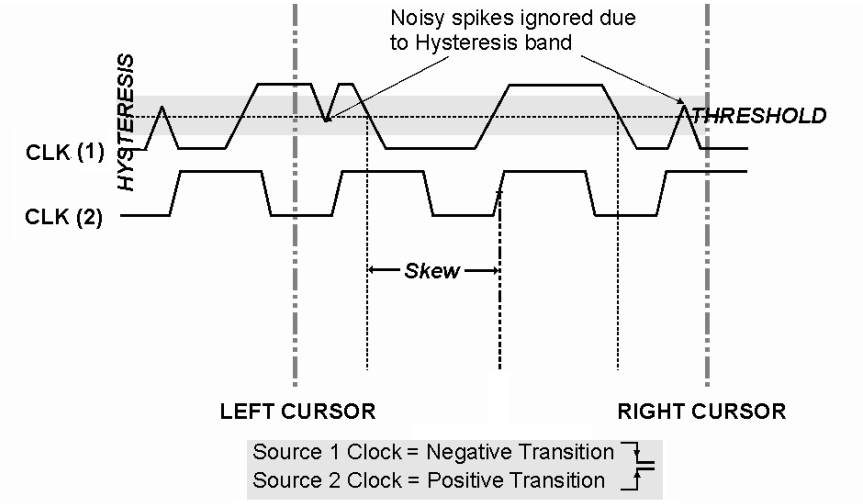
Time-parameter estimation depends on the number of cycles included within the observation window. If the number of cycles is not an integer, parameter measurements such as rms or mean will be biased. However, only the last value is actually displayed, the mean being available when statistics are enabled. To avoid these bias effects, cyclic parameters can be chosen, including crms and cmean, that restrict the calculation to an integer number of cycles.



Determining Differential Time Measurements

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

Parameters such as Skew require the transition polarity of the clock and data signals to be specified. A hysteresis range may be specified to ignore any spurious transition that does not exceed the boundaries of the hysteresis interval. In the figure below, Skew measures the time interval separating the falling edge of Source1 from the rising edge of Source2.



Math

Math traces (F*x*) 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. It can even be applied to another math trace, allowing you to chain operations (for example, trace F1 can show the average of C1, while trace F2 provides the integral of F1). 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 instrument, enhanced math analysis tools customized for various industries and applications are offered through optional software packages. To learn about math tools available in each optional package, see the product datasheets at teledynelecroy.com.

If you have installed software options, the new capabilities are usually accessed through the Analysis menu, rather than the Math menu, although special measure parameters and math functions will be available when using Measure and Math dialogs.

Math Function Set Up

Use the Function dialog to set up math function traces. Math functions take as input one or more channel, zoom, memory or math traces and output a new function trace (F*x*). Any additional settings required for the operator will appear on a subdialog at the right of the screen.

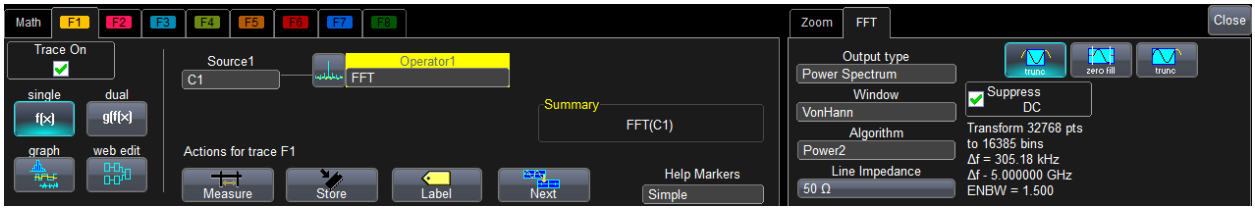
Single functions perform one operation on one or two input sources.

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

Setting Up New Functions

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

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



3. Choose a **single f(x)** or **dual g(f(x)** operator function.

You can also choose Web Edit, in which case this function is added as a terminal to the processing web. See the instructions for Using Wed Edit to set up the function.

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

   Note: On instruments with the Advanced Customization (XDEV) option, the Operator can be a custom script written in one of several scripting "languages." Choose from the Custom submenu and import or write the script in the Script Editor. See Using PScript for a description of the process.

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 subdialog of the same name as the operator. Touch the tab to open the dialog and make any further settings. These are explained on the dialog.

7. If you're creating a dual function, repeat the procedure for the second operator.

Graphing

The **Graph button** on the Function (F*x*) dialogs allows you to simultaneously apply a measurement parameter to the function trace and plot the results using histogram, track, or trend. See Histogramming and Track and Trend.

Tip: The plots are the same as those created using the toolbar on the Parameter (P*x*) dialog.

As with other math functions, configurable settings will appear on subdialogs after the plot is selected.

Adjusting Memory or Math Traces

Unlike channel traces, the scale of memory (M*x*) or math function (F*x*) traces can be adjusted directly without having to create a separate zoom trace. The same set of zoom factor controls used for zoom traces appear on the **Zoom subdialog**, but in this context they only rescale the active math or memory trace rather than create a new zoom. This applies to any trace that is created as a math function (F*x*) trace, including traces generated through analysis options and graphs.

You can, if you wish, create a separate zoom trace from a memory or function trace the same as you would normally create a zoom (draw a selection box, Add New Zoom, etc.). In this case, you choose one of the zoom locations (Z*x*) in which to draw the trace, but the source trace remains at the original scale.

Math Dialog

Once a math function has been created and saved on the Function (F*x*) dialog, use the main Math dialog to quickly enable/disable it. You can also use this dialog to quickly turn on/off zoom traces.



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

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

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

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

Average Function

Setting Up Averaging

To apply Continuous or Summed Averaging as a Math function:

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

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

Summed Averaging

Summed Averaging is the repeated addition, with equal weight, of successive source waveform records. If a stable trigger is available, the resulting average has a random noise component lower than that of a single-shot record. Whenever the maximum number of sweeps is reached, the averaging process stops. In Summed averaging, you specify the number of acquisitions to be averaged. The averaged data is updated at regular intervals.

An even larger number of records can be accumulated simply by changing the number in the dialog. However, the other parameters must be left unchanged or a new averaging calculation will be started. You can pause the averaging by changing the trigger mode from NORMAL/AUTO to STOP. The instrument resumes averaging when you change the trigger mode back to NORMAL/AUTO.

You can reset the accumulated average by pushing the CLEAR SWEEPS button or by changing an acquisition parameter such as input gain, offset, coupling, trigger condition, timebase, or bandwidth limit. The number of current averaged waveforms of the function, or its zoom, is shown in the acquisition status dialog. When summed averaging is performed, the display is updated at a reduced rate to increase the averaging speed (points and events per second).

Continuous Averaging

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

You determine the importance of new data vs. old data by assigning a weighting factor. Continuous averaging allows you to make adjustments to a system under test and to see the results immediately. The formula for both summed and continuous averaging is:

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

However, by setting a Sweeps value, you establish a fixed weight that is assigned to the old average once the number of sweeps is reached. For example, for a sweeps (weight) value of 4:

Sweep	New Average =
1 (no old average yet)	(new data +0 * old average)/(0 + 1) = new data only
2	(new data + 1*old average)/(1 + 1) = 1/2 new data +1/2 old average
3	(new data + 2 * old average)/(2 + 1) = 1/3 new data + 2/3 old average
4	(new data + 3 * old average)/(3 + 1) = 1/4 new data + 3/4 old average
5	(new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average
6	(new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average
7	(new data + 4 * old average)/(4 + 1) = 1/5 new data + 4/5 old average

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.

Copy Function

The Copy math function saves a copy of your present waveform in its unprocessed state to the first available memory location. While processing may continue on the original waveform, the copy enables faster throughput in some cases by preserving the original data. No calculations need to be undone on the copy before additional math can be calculated. This benefit of faster throughput, however, comes at the expense of memory usage.

Follow the ususal steps to set up a math function, selecting Copy from the Misc submenu.

On the Wform Copy subdialog, optionally Reset Count or Change BatchSize.

ERes Function

ERes (Enhanced Resolution) filtering increases vertical resolution, allowing you to distinguish closely spaced voltage levels. The instrument's ERes function is similar to smoothing the signal with a simple, moving-average filter. However, it is more efficient concerning bandwidth and pass-band filtering.

Use ERes:

- On single-shot acquisitions, or where the data record is slowly repetitive (cases where you cannot use averaging).
- To reduce noise on noticeably noisy signals when you do not need to perform noise measurements.
- When performing high-precision voltage measurements (e.g., zooming with high vertical gain).

Setting Up ERes

To apply ERes as a Math function:

1. Follow the usual steps to set up a math function, selecting **Eres** from the **Filter** submenu.
2. Touch the **Trace On** checkbox.
3. On the **Eres** subdialog, then touch **bits** and make a selection from the pop-up menu.



Tip: To quickly set up ERes as a pre-processing function, access the channel setup dialog (C*x*) and select a **Noise Filter (ERes)** bit size.

How the Instrument Enhances Resolution

The instrument's enhanced resolution feature improves vertical resolution by a fixed amount for each filter. This real increase in resolution occurs whether or not the signal is noisy, or whether it is single-shot or repetitive. The signal-to-noise ratio (SNR) improvement depends on the form of the noise in the original signal. The enhanced resolution filtering decreases the bandwidth of the signal, filtering out some of the noise.

The instrument's constant phase finite impulse response (FIR) filters provide fast computation, excellent step response in 0.5 bit steps, and minimum bandwidth reduction for resolution improvements of between 0.5 and 3 bits. Each step corresponds to a bandwidth reduction factor of two, allowing easy control of the bandwidth resolution trade-off.

Resolution increased by	-3 dB Bandwidth (x Nyquist)	Filter Length (Samples)
0.5	0.5	2
1.0	0.241	5
1.5	0.121	10
2.0	0.058	24
2.5	0.029	51
3.0	0.016	117

With low-pass filters, the actual SNR increase obtained in any particular situation depends on the power spectral density of the noise on the signal.

The improvement in SNR corresponds to the improvement in resolution if the noise in the signal is white (evenly distributed across the frequency spectrum). If the noise power is biased towards high frequencies, the SNR improvement will be better than the resolution improvement.

The opposite may be true if the noise is mostly at lower frequencies. SNR improvement due to the removal of coherent noise signals—feed-through of clock signals, for example—is determined by the fall of the dominant frequency components of the signal in the passband. This is easily ascertained using spectral analysis. The filters have a precisely constant zero-phase response. This has two benefits. First, the filters do not distort the relative position of different events in the waveform, even if the events' frequency content is different. Second, because the waveforms are stored, the delay normally associated with filtering (between the input and output waveforms) can be exactly compensated during the computation of the filtered waveform.

The filters have been given exact unity gain at low frequency. ERes should therefore not cause overflow if the source data is not overflowed. If part of the source trace were to overflow, filtering would be allowed, but the results in the vicinity of the overflowed data—the filter impulse response length—would be incorrect. This is because in some circumstances an overflow may be a spike of only one or two samples, and the energy in this spike may not be enough to significantly affect the results. It would then be undesirable to disallow the whole trace.



Note: While ERes improves the resolution of a trace, it cannot improve the accuracy or linearity of the original quantization. The pass-band causes signal attenuation for signals near the cut-off frequency. The highest frequencies passed may be slightly attenuated. Perform the filtering on finite record lengths. Data is lost at the start and end of the waveform and the trace ends up slightly shorter after filtering. The number of samples lost is exactly equal to the length of the impulse response of the filter used: between 2 and 117 samples. Normally this loss (just 0.2 % of a 50,000 point trace) is not noticed. However, you might filter a record so short that no data is output. In that case, however, the instrument would not allow you to use the ERes feature.

Fast Wave Port Function

Fast Wave Port is a function of Teledyne LeCroy X-Stream oscilloscopes that enables you to insert a custom algorithm written in the C/C++ language into the oscilloscope's processing stream. It also maximizes data throughput from the acquisition system to your custom algorithm for purposes of debugging. Custom algorithms can be written right on the oscilloscope using the Script Editor.

This is possible because two Windows system processes may share a region of memory, enabling high-speed data transfer between the acquisition system and the custom algorithm, which runs in a separate process from the oscilloscope application. A major benefit of Fast Wave Port is that your algorithm may be implemented and, more importantly, debugged independently of the main application.

It is important to note that transferring the results of your algorithm back into the X-Stream processing stream is optional. If performance is the primary goal, and further processing of the results within the X-Stream software is not required, this step may be skipped.

Fast Wave Port was designed for use with the C/C++ programming language. However, it is theoretically possible for the processing to be implemented in any language supporting Windows named events (Mutex) that can open a named memory-mapped file. No guarantee can be given, however, as to the behavior of the function using anything but C/C++.

The size of the memory window is fixed at 80 MB, equating to 40 ms.

See the Fast Wave Port topic in the oscilloscope Help for an example C++ application.

To use Fast Wave Port, follow the usual steps to set up a math function, selecting **Fast Wave Port** as the Operator, then make the appropriate settings on the **Fast Wave Port** subdialog.

The **Port Name** is critical, indicating the base name of the memory window and the global objects within the Windows O/S. Only change the default value if using multiple Fast Wave Port functions in parallel.



Note: The base name must match the base name used in the client application.

The full names of the default global objects are as follows:

Object	Full Name
Memory Mapped File	FastWavePort1File
Data Available Event	FastWavePort1MutexDataAvailable
Processing Complete Event	FastWavePort1MutexProcessingComplete

The **Timeout** control specifies the amount of time the DSO waits for the custom processing function to complete. This prevents the DSO from waiting indefinitely for a custom processing function which may never arrive. Be careful to set this control to a reasonable value, meaning a time longer by a reasonable margin than the custom processing is ever expected to take.

FFT Function

For a large class of signals, you can gain greater insight by looking at spectral representation rather than time description. Signals encountered in the frequency response of amplifiers, oscillator phase noise and those in mechanical vibration analysis, for example, are easier to observe in the frequency domain.

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

While FFT has become a popular analysis tool, some care must be taken with it. In most instances, incorrect positioning of the signal within the display grid will significantly alter the spectrum, producing effects such as leakage and aliasing that distort the spectrum.

An effective way to reduce these effects is to maximize the acquisition record length. Record length directly conditions the effective sampling rate and therefore determines the frequency resolution and span at which spectral analysis can be carried out.

Setting Up FFT

1. Follow the usual steps to set up a math function, selecting **FFT** from the **Frequency Analysis** submenu.
2. Open the **FFT** subdialog.
3. Choose an **Output type**.
4. If your Output Type is Power Density or Power Spectrum, also enter **Line Impedence**. By default, the FFT function assumes a termination of 50 Ohms. If an external terminator is being used, this setting can be changed to properly calculate the FFT based on the new termination value.

   If your Output type is Phase, enter a **Group Delay Shift** value.
5. Optionally, choose a weighting **Window** (see below).
6. Check the **Suppress DC** box to make the DC bin go to zero. Otherwise, leave it unchecked.

Choosing a Window

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	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)	Reduces leakage and improves amplitude accuracy. However, frequency resolution is also reduced.
Hamming	Reduces leakage and improves amplitude accuracy. However, frequency resolution is also reduced.
Flat Top	Provides excellent amplitude accuracy with moderate reduction of leakage, but with reduced frequency resolution.
Blackman-Harris	Reduces leakage to a minimum, but with reduced frequency resolution.

FFT Window Filter Parameters				
Window Type	Highest Side Lobe (dB)	Scallop Loss (dB)	ENBW (bins)	Coherent Gain (dB)
Rectangular	-13	3.92	1.0	0.0
Von Hann	-32	1.42	1.5	-6.02
Hamming	-43	1.78	1.37	-5.35
Flat Top	-44	0.01	3.43	-11.05
Blackman-Harris	-67	1.13	1.71	-7.53

Interpolate Function

Linear interpolation, which inserts a straight line between sample points, is best used to reconstruct straight-edged signals such as square waves. This is the default interpolation method used by the oscilloscope.

(Sinx)/x interpolation, on the other hand, is suitable for reconstructing curved or irregular waveshapes, especially when the sampling rate is 3 to 5 times the system bandwidth.

Cubic interpolation can be used to create a smooth, continuous function by applying a third-degree polynomial.

For each method, you can select an upsample factor of 2 to 50 points to insert between samples.

1. Follow the usual steps to set up a math function, selecting **Interpolate** from the **Filter** submenu.
2. Touch **Algorithm** and select an interpolation type.
3. Enter the **Upsample** factor by which to increase sampling.

   You can use the Up and Down buttons to set this value. To make fine increment changes, deselect the "only by 2, 5..." checkbox.
4. In **Half Width** enter the width in taps of the filter.
5. To use a weighting function rather than an upsample factor, select it from **Weighting**. See FFT for more information about the functions. If using the Kaiser function, also enter the **Beta** value.

Sparse Function

The Sparse math function allows you to thin out an incoming waveform by skipping points at regular intervals, and by starting acquisition at a particular offset (point). The **Sparsing factor** specifies the number of sample points to reduce the input waveform by. A sparsing factor of 4, for example, says to retain only one out of every 4 samples. A **Sparsing offset** of 3, on the other hand, says to begin on the third sample, then skip the number of samples specified by the sparsing factor (4). In this way, the sample rate is effectively reduced.

For the sparsing factor (interval), you can set a value from 1 to 1,000,000 points. For the sparsing offset you can set a value from 0 to 999,999.

1. Follow the usual steps to set up a math function, selecting **Sparse** from the **Misc** submenu.
2. Touch the **Sparsing factor** control and provide a Bandwidth Limit value.
3. Touch the **Sparsing offset** control and provide a value.

Rescale Function and Assigning Units

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



Tip: Analog channel traces may be rescaled directly using the C*x* dialogs. They do not require a separate Math function.

Setting Up Rescaling

1. Follow the usual steps to set up a math function, selecting **Rescale** from the **Functions** submenu.
2. Touch the **Rescale** subdialog tab.
3. To 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 new unit of measure.

   You can combine units following these rules:
   - For the quotient of two units, use the character "**:/**"
   - For the product of two units, use the character "**.**"
   - For exponents, append the digit to the unit without a space (e.g., "S2" for seconds squared)

   Note: Some units are converted to simple units (e.g., V.A becomes 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		

Phistogram Function

The Phistogram function creates a histogram of the samples within a vertical or horizontal "slice" of a persistence map to reveal the features that are only visible when several acquisitions have been superimposed on one another. Vertically, each bin contains a class of similar amplitude levels; horizontally, each bin contains a class of similar time values.

The source trace must have Persistence "on" to generate the persistence map. Eye diagrams, which are a type of persistence map, are a an especially useful class of traces to which you can apply the Phistogram function.

For a histogram of...	Use...
A crossover point in time or amplitude on an eye diagram *	Vertical and horizontal slice persistence histogram
Cumulative jitter on an eye diagram *	Horizontal slice persistence histogram
Signal-to-noise ratio on an eye diagram *	Vertical slice persistence histogram

* Requires optional software package capable of creating eye diagrams, such as SDA II orSerial Decoder TDME software.

1. Follow the usual steps to set up a math function, choosing a **Single** operator function using the **Phistogram** operator.
2. On the **Phistogram** subdialog, choose the slice of the persistence map to histogram:
   - **Slice Direction** of determines whether the slice transverses the entire height of the map centered around a time (vertical), or transverses a period of time in the map centered around a level (horizontal).
   - **Slice Center** represents the center time or level of the slice.
   - **Slice Width** represents the total time or amplitude of the slice, half each side of the center.

     Note: This operation does not affect, and is not affected by, Center and Width settings made on any existing histograms.
3. Choose **Center Cursor** to place a measurement cursor at the center of the slice.

Ptrace Functions

The Ptrace Mean, Ptrace Range, and Ptrace Sigma math functions (found in the Jitter Functions sub-menu of the Math Operator selector) compute a vector trace based on a persistence map of underlying signal acquisitions without destroying the underlying data, allowing the display of analytical results from raw data. Further measurements or Pass/Fail testing can be performed on the resulting function traces.

The function source trace must have Persistence "on" to generate the persistence map.

Function	Options	Methodology	Uses
Ptrace Mean		For each vertical time slice on the persistence map, Ptrace Mean calculates and plots a trace corresponding to the map's mean value. Single-shot signals sampled at or above 2 GS/s and accumulated in the persistence map can be traced at a resolution of 10 ps (100 GS/s equivalent sampling). The persistence trace average can be further analyzed using the instrument's standard parameters, such as rise time.	See edge detail in a fast signal or eliminate noise on a persistence map.
Ptrace Range	**% population range**. A percentage of the population of the persistence map can be chosen from which the envelope will be formed, enabling exclusion of infrequent events (artifacts).	For each vertical time slice on the persistence map, Ptrace Range calculates and plots an envelope corresponding to the map's range. The range can then be used in further processing: for example, as a source for Pass/Fail masks.	Assess typical noise on a persistence map.
Ptrace Sigma	**Scale to standard deviations**. This allows you to select a sigma from 0.5 to 10.0, which expands those parts of the sigma envelope representing waveform regions. This is useful for making a tolerance mask.	For each vertical time slice on the persistence map, Ptrace Sigma calculates and plots an envelope corresponding to the map's standard deviation. Multiples of sigma can also be done using sigma. The sigma can be used in further processing; for example, as a source for Pass/Fail masks.	Assess worst case noise on a persistence map and use it to create a tolerance mask for Pass/Fail testing.

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.
Copy	Copies waveform in its unprocessed state to the first available memory location.
Correlation	Calculates a measure of similarity of two waveforms, or a waveform against itself, as a function of a time-lag applied to one of them.
Derivative	Calculates the derivative of adjacent samples using the formula: *(next sample value – current sample value) / (horizontal sample interval)*
Deskew	Shifts trace in time the amount of the deskew factor.
Difference	For every point in the waveform, 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.
Exp	Calculates the antilog to the base e of the source; that is, e raised to the power equal to the source.
Exp10	Same as Exp, using base 10.
FFT	Computes a frequency spectrum with optional Rectangular, Von Hann, Flat Topp, Hamming, Blackman-Harris, and Hanning windows. Calculates up to 128 Mpts. Also allows FFT Averaging through use of a second math operator. See FFT.
Floor	Calculates the lowest vertical values of a waveform at each horizontal value for a specified number of sweeps.
Histogram	Plots the number of data points that fall into statistically significant intervals or bins. Bar height relates to the frequency at which data points fall into each interval/bin.
Integral	Calculates the linearly rescaled integral (with multiplier and adder) of a waveform input starting from the left edge of the screen using the formula: *(current sample value + next sample value) * (horizontal sample interval)* Each calculated area is summed with the previous sum of areas. The multiplier and adder are applied before the integration function.
Interpolate	Inserts points between sampled points (upsamples) according to one of three algorithms: Linear (straight line), Sinx/x (curved), and Cubic (spine). Interpolation factor of 2 to 50 determines number of points in the upsample.
Invert	For every point in the waveform, the inverse of that point is calculated.

Operator	Definition
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.
MatLab math	Produces a waveform using a custom MatLab function. Requires XDEV option to edit functions through the oscilloscope GUI using MatLab Script.
Phistogram	Creates a histogram based on the displayed pixels of a persistence map falling within a user-defined vertical and/or horizontal box (slice). The source trace must have Persistence turned "on."
Product	For every point in the waveform, the value of Source1 is multiplied by the value of Source 2. Source1 and Source2 must have the same horizontal units and scale.
Ptrace mean	Plots the mean value of each sample point in a persistence map. The source trace must have Persistence turned "on."
Ptrace range	Generates a waveform with a width derived from the population range of a persistence map. The source trace must have Persistence turned "on."
Ptrace sigma	Generates a waveform with a width derived from the sigma (sum) of a persistence map. The source trace must have Persistence turned "on."
Ratio	For every point in the waveform, 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.
Roof	Calculates the highest vertical values of a waveform at each horizontal value for a specified number of sweeps.
Segment	Selects one segment from a source waveform to place in a sequence waveform. Used in Sequence sampling mode.
Sinx/x	Performs 10-to-1 interpolation using a Sin(x)/x filter.
Sparse	“Thins,” or decimates, an incoming acquisition by dropping sample points at regular intervals. Sparsing factor specifies the number of points to drop between retained samples (e.g., factor of 4 retains 1 then drops 4). Sparsing offset specifies the point at which to begin applying the sparsing factor (e.g., offset of 3 begins count on the third sample (3), then drops the number of samples specified by the sparsing factor (4).
Square	For every point in the waveform, 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.
Track	Generates a waveform composed of parameter measurements that is time synchronous with the source waveform. The vertical units are those of the source parameter value and the horizontal units are seconds. Parameter values are posted at the sampling rate.
Trend	Produces a waveform composed of a series of parameter measurements in the order the measurements were taken. The vertical units are those of the source parameter, the horizontal unit is measurement number. The trend contains a single value for each measurement.
Zoom	Produces a magnified trace of a selected portion of the input waveform. See Zooming Traces.

Memory

The instrument is equipped with four internal memory slots (M*x*) to which you can copy any waveform that is active on the grid. This is a convenient way to store an acquisition for later viewing and analysis.

Memories can be used as source inputs for most oscilloscope math and measurements, allowing you to compare historical data to a live acquisition or perform "what if" modeling on saved acquisitions.

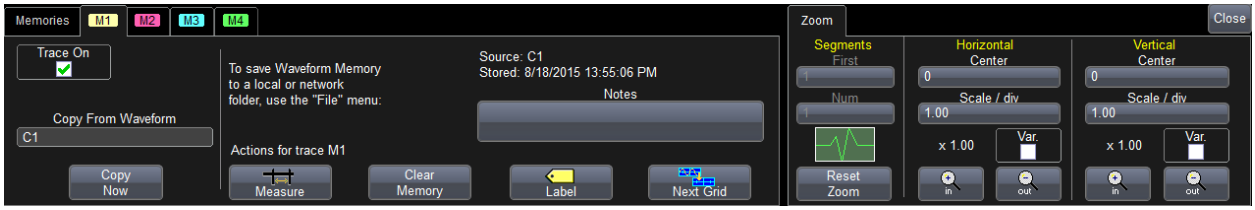
Saving Memories

Store memories on the Memory dialogs (M*x*). Memories are created at the same scale as the source trace, but they can be adjusted independently by using the zoom factor controls that appear next to the M*x* dialogs.

Save Waveform to Memory

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

1. 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.



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

Save (External) Waveform Files to Memory

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

Restoring Memories

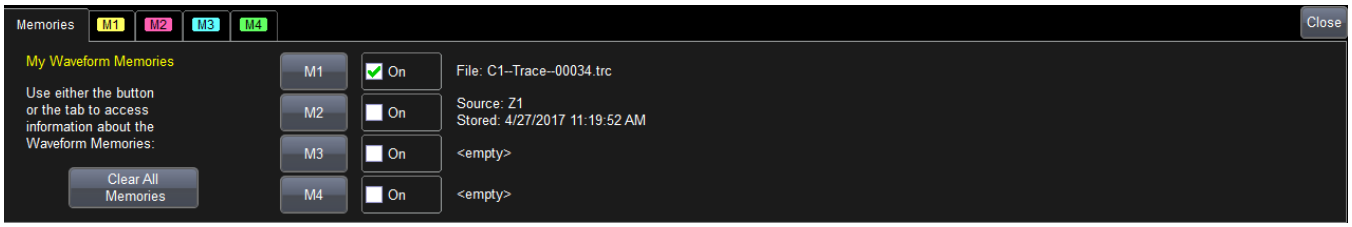
The Memories dialog is a convenient panel for restoring saved memories to the display.

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

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

Touch **Clear All Memories** to empty the memory banks.

Caution: Memories cannot be restored once they have been erased.



Analysis Tools

The Analysis menu tools complement the standard math and measurements to help you understand the behavior of waveforms.

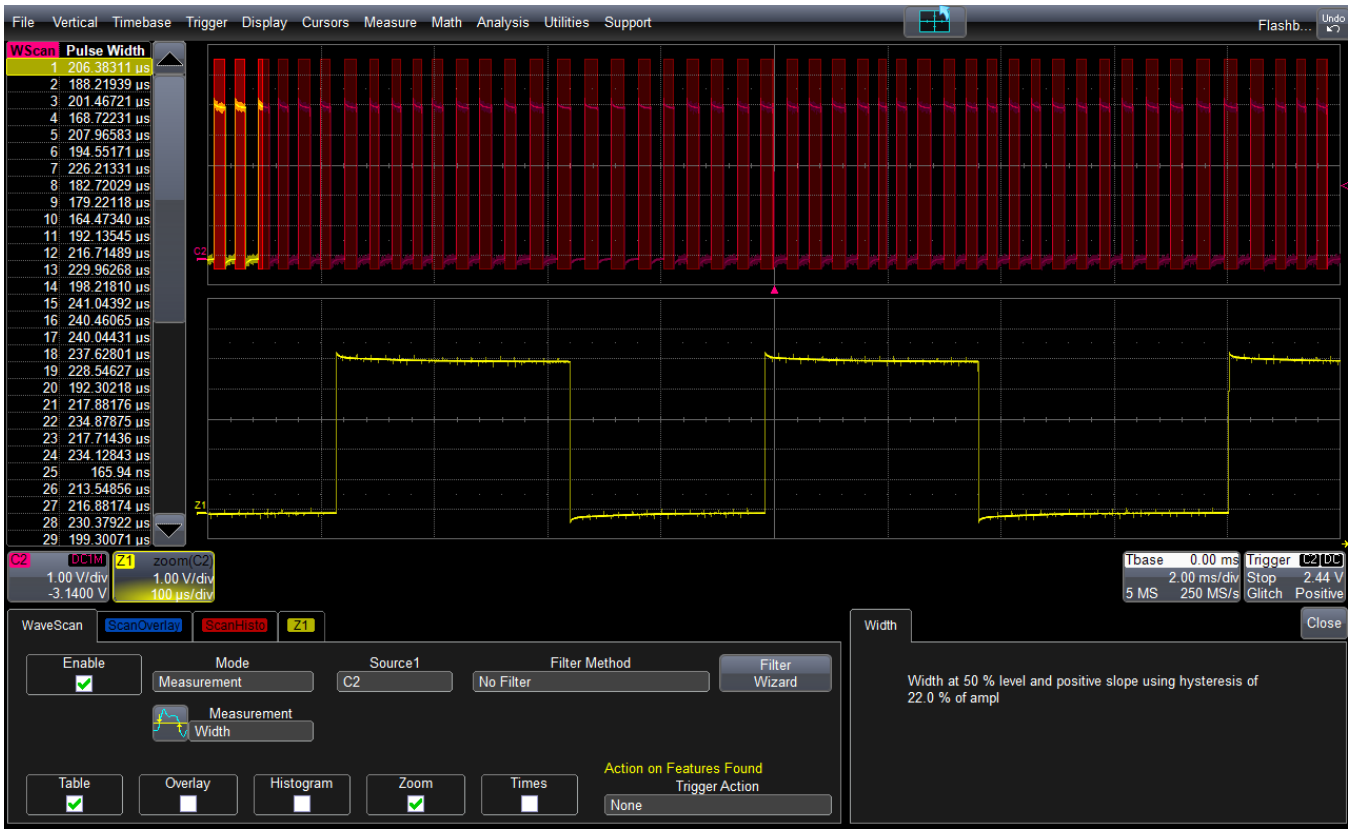
WaveScan searches single or multiple acquisitions for events that meet specific criteria.

Pass/Fail Testing shows whether waveforms or measurements meet .

Optional software packages may be purchased for specialized uses, such as power analysis. In most cases, these options are added to the Analysis menu. Documentation for software options can be found at teledynelecroy.com/support/techlib under Manuals > Software Options. In addition, documentation for many options available on your platform can be found in the online Help under the Support menu.

WaveScan

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



WaveScan window with different scan "views" turned on.

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

Note: The instrument reverts to Real-time sampling mode when WaveScan is enabled.

Setting Up WaveScan

Set up your source channel and triggers before setting up the scan.

1. Press the front panel **Stop button** to stop acquisition.
2. Choose **Analysis > WaveScan**.
3. Check **Enable**.
4. Choose the **Source** waveform.
5. Choose the Scan Mode and enter values for any additional settings that appear at the right of the dialog based on your selection.
6. Select each Scan View in which you wish to display results by checking the box at the bottom of the dialog. Each view selected is displayed simultaneously.
7. If you're using Scan Overlay view, on the Scan Overlay dialog **Clear Sweeps**. If desired, set up the Persistence display.
8. Optionally, choose an **Action** to trigger when an event that meets your scan criteria is found.
9. Restart acquisition.

Scan Modes

The Scan Mode determines the type of search to be performed. Select the **Mode** along with the **Source** trace to be searched on the main WaveScan dialog. For each mode, different controls appear on the WaveScan dialog, providing additional inputs to the search criteria. Make the appropriate entries in these fields before starting the search.

Edge Mode

Edge Mode is used for detecting the occurrence of edges. Events that meet the threshold level are captured and tabulated. When the acquisition is stopped, scan filters can be applied to the edges to find specific characteristics. Edge Mode settings are:

- **Slope**. Choose Pos, Neg, or Both.
- **Level is** (set in...). Choose Percent or Absolute.
- **Percent/Absolute Level**. Enter a threshold value as a percentage of Top to Base or voltage level.

Non-monotonic Mode

Non-monotonic Mode looks for edges that cross a threshold more than once between high and low levels. All events that meet the criteria of slope, hysteresis, and level are presented in a table and highlighted in the source trace. The value displayed in the table is the difference of the max. and min. of the non-monotonicity. This can be confirmed with cursors. The hysteresis value is used to eliminate noise. A non-monotonicity is detected only when its amplitude is greater than the hysteresis. Therefore, when setting a hysteresis level, set a value that is greater than the amplitude of the noise. Settings are:

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

Runt Mode

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

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

- Runt Type. Choose Both, Pos, or Neg.
- Hysteresis. Enter the hysteresis level as a percentage or voltage.
- Low/High Threshold. Enter the levels as a percentage or voltage.
- Absolute Levels. Check this box to enter levels as absolute voltage instead of percentage.

Measurement Mode

Measurement Mode is used for applying filters to measurements to find those that meet your defined criteria, helping to isolate particular events within many samples. Markers appear over the source trace to indicate the location of measurement, while the table displays values for the selected parameter that meet the criteria. Measurement Mode settings are:

- Measurement. Choose the measurement parameter you wish to search.
- Filter Method. Choose the operator that indicates the desired relationship to the Filter Limit. Only measurements that meet this criteria are returned.
- Filter Limit. Enter the value that completes the filter criteria.

Alternatively, you can use the Filter Wizard to create the filter criteria.

Serial Pattern Mode

Serial Pattern Mode is used for finding 2- to 64-bit patterns in digital sequences; ideal for bursted patterns where a PLL cannot lock. Serial Pattern Mode settings are:

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

Bus Pattern Mode

Bus Pattern Mode (only) is used for finding 2- to 16-bit patterns across the digital lines. Bus Pattern Mode settings are:

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

Scan Views

Scan Views are different ways to view your WaveScan results. Just check the boxes at the bottom of the WaveScan dialog for those views you wish to display simultaneously.

Note: The number of grids varies depending on which views are enabled. WaveScan handles this function automatically, and you cannot move traces among grids as in normal operation.

Additional controls for Scan Overlay and Zoom views are on their respective dialogs. If you turn on these traces from those dialogs, you must turn them off from there, too.

Source Trace

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

Table

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

Scan Overlay

Scan Overlay view uses colored overlays to show the location of events meeting the criteria.

Zoom

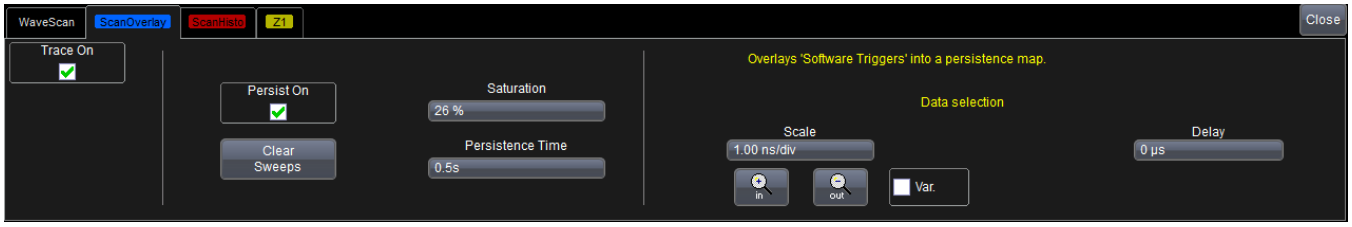
Zoom view works exactly as it does elsewhere in the X-Stream software, creating a new trace that is a magnified section of the source trace. A Z*x* tab appears by default when you launch WaveScan; see zoom factor controls for an explanation of the remainder of the controls found on this dialog.

Scan Overlay

To apply monochromatic persistence to a Scan Overlay:

1. Select **Scan Overlay** when setting up the wave scan, then open the **Scan Overlay dialog**.
2. Check **Persistence On**.
3. Enter a **Saturation** level as a percentage. All samples above the saturation level are assigned the highest color intensity.
4. Choose a **Persistence Time**. The higher the time, the more static the persistence display.

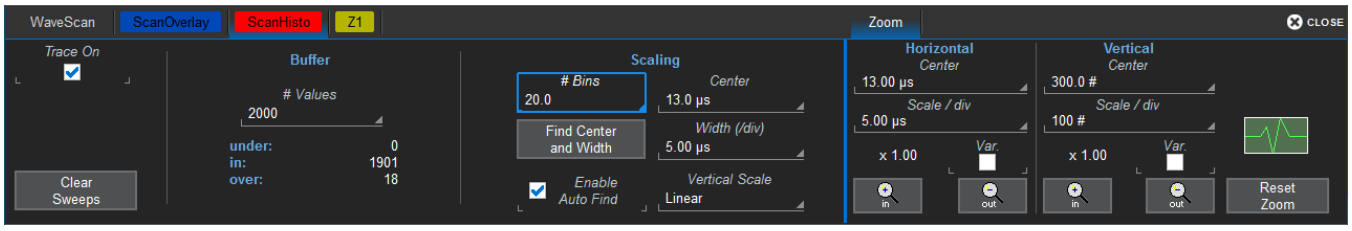
To adjust the scan overlay to "zoom" in or out: touch the **In/Out buttons**, or touch **Scale** and enter new values. Check **Var.** to adjust scale in finer steps than the default 1, 2, 5.



Scan Histogram

Scan Histogram is an additional WaveScan "view" that generates a histogram to give you a statistical view of edges that meet your search criteria. Set it up as you would any histogram:

1. Enter the total **#Values** in the histogram buffer.
2. Enter the **#Bins** (bars) in the histogram.
3. Enter the **Center** value and **Width** of each bin from center, or check **Enable Auto Find** and let the sofware **Find Center and Width**.
4. Choose a **Vertical Scale** method:
   - Linear allows the histogram to build vertically as data accumulates. When the histogram reaches the top of the display it rescales the vertical axis to keep it on screen.
   - LinConstMax keeps the histogram at near full scale and rescales the vertical axis as data is accumulated.



WaveScan Search

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

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

   Or

   If you know the WaveScan table index (row) number of the event you wish to find, enter it in Idx.

Pass/Fail Testing

Pass/Fail testing allows you to define a set of qualifying conditions that an acquisition may "pass" or "fail" when tested against, then take actions depending on whether the result is a pass or a fail.

- Mask testing compares sampled values to a pre-defined area of the grid (the "mask") to see if they fall within or without.
- Parameter comparison compares a measurement result to a pre-defined value (Param compare) or to another measurement (Dual param compare).

You can preconfigure eight qualifiers (Q*x*) using either method, which are then enabled by selecting them on the Pass/Fail dialog and defining what results constitute a "Pass" on the Actions dialog.

Mask Test Qualifiers

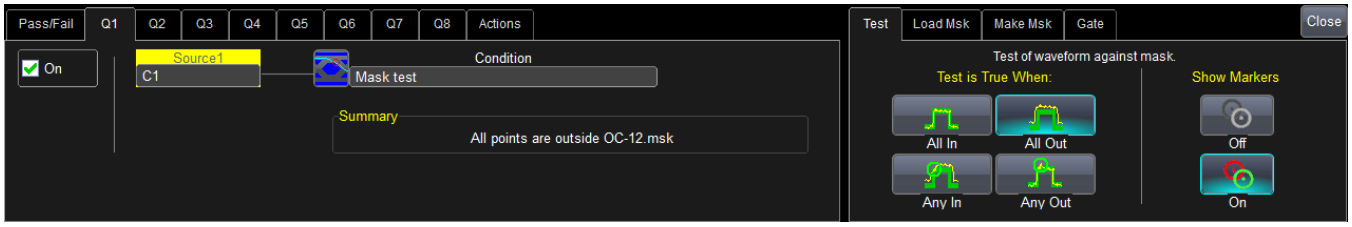
A mask defines an area of the grid against which a source Channel, Zoom, or Math trace is compared. Test conditions are associated with the mask, defining how the waveform is to be compared to the masked area (e.g., some/all values fall within, some/all values fall outside), and a pass or fail result is returned indicating the condition was found to be true or false.

Mask testing can be done using a pre-defined mask or a mask created from a waveform with user-defined vertical and horizontal tolerances. Some industry standard masks used for compliance testing are included with the oscilloscope software. The mask test can be confined to just a portion of the trace by the use of a measure gate.

Access Mask Test Dialogs

1. Choose **Analysis > Pass/Fail** to display the **Pass/Fail** dialog.
2. Touch the **Q*x* button** where you want to set up the qualifier.
3. Select **Mask test**.

The Q*x* dialog opens with the Mask test condition selected and the **Test**, **Load Mask**, **Make Mask**, and **Gate** subdialogs displayed. On these dialogs, you manage, make, and apply gates to your mask.



Make Mask

Use this procedure to create a new mask based on a live waveform. The mask covers the area of the waveform plus the boundaries you enter.

1. Open the **Make Mask** subdialog.
2. If desired, enter a new **Destination File Name** and path, or touch Browse and select a previous file to overwrite. The file name should end with the **.msk** extension.
3. Touch the **Ver Delta** and **Hor Delta** fields and enter boundary values using the pop-up Virtual Keypad or the front panel Adjust knob.
4. Touch **Make from Trace**.

Load Mask

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

1. Open the **Load Mask** subdialog.
2. To use a saved .msk file, touch **File** and select the mask.

   To use an industry standard mask, touch **Standard** and select the mask from the pop-up.
3. Check **View Mask** to display the mask over the trace.

Remove a Mask from the Display

Touch the **Delete All Masks** button on the Load Mask subdialog. Alternatively, open the **Qx** dialog where a particular mask is set, and on the **Load Mask** subdialog, touch **Delete**.

Set Gates

Set gates to limit the portion of the waveform that is compared to the mask.

1. Open the **Gate** subdialog.
2. Enter the **Start** and **Stop** horizontal divisions that mark the 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.

   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 "True"

1. Open the **Test** subdialog.
2. Select one of the conditions that, when true (yes), results in a pass.
3. Optionally, turn **On** markers to show where on the waveform mask violations have occurred.

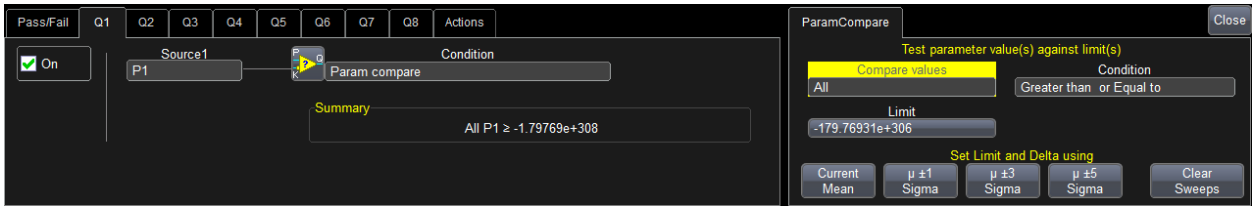
Param(eter) Compare Qualifiers

Pass/Fail qualifiers (Q*x*) can be configured to compare different parameter measurements (P*x*) to each other or to a user-defined limit (or statistical range). The Summary field shows the logic represented by the qualifier. Check it to ensure your selections are producing the test you want.

Comparing a Measurement to a Limit

This method compares the result of a selected parameter to a user-defined limit or statistical range.

1. Choose **Analysis > Pass/Fail** to display the **Pass/Fail** dialog.
2. Touch the **Q*x* button** where you want to set up a qualifier.
3. From the pop-up menu, choose **Param compare**.
4. On the Q*x* dialog, touch **Source1** and choose the parameter.



5. On the ParamCompare subdialog, touch **Compare Values** and select **All** (every value must be within limit to be "true") or **Any** (any value can be within limit to be "true").
6. Touch **Condition** and select a math operator, then touch **Limit** and enter the value that completes the condition. The value entered acquires properties corresponding to the parameters being tested. For example, if you are testing a time parameter, the unit is seconds.
7. If you chose to set a Delta limit, also enter the **Absolute or % Delta** value. You may choose instead to set Limit and Delta using one of the buttons at the bottom of the dialog.

Comparing Parameter Measurements

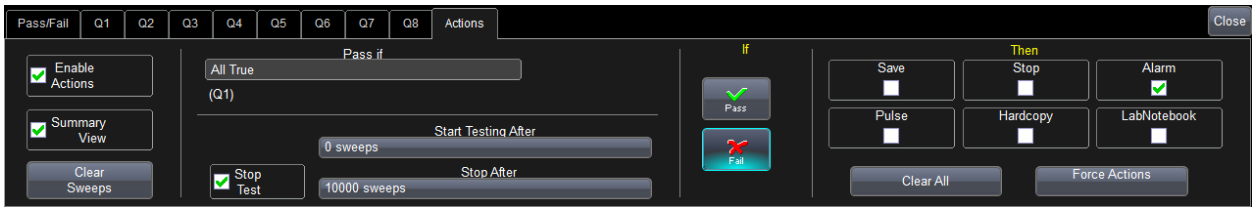
This method compares one parameter to another, rather than a limit.

1. Choose **Analysis > Pass/Fail** to display the **Pass/Fail** dialog.
2. Touch the **Q*x* button** where you want to set up a qualifier.
3. From the Pass/Fail Condition menu, choose **Dual Param Compare**.
4. On the left-hand Qx dialog, select the parameters to compare in **Source1** and **Source2**.
5. On the ParamCompare subdialog, choose to **Compare All** (every value must meet condition to be "true") or **Any** (any value can meet condition to be "true").
6. Touch **Condition** and select the operator that expresses the relationship between parameters (e.g., *All* P1 values must be *less than* P2).

Define Pass/Fail Tests

The Qualifiers define the conditions that would result in "true" but don't in themselves determine the test result. You must separately define what constitutes a "pass" or a "fail" on the Actions dialog. A Pass or Fail result can be made to produce various additional actions, such as sending a pulse to another device.

1. After setting up the Pass/Fail test qualifiers, open the **Actions tab**.



2. Optionally, check **Summary View** to see a running summary of results over the total number of sweeps.
3. Select the test Pass criteria in **Pass If**.
4. To apply additional actions to the test result:
   - Check **Enable Actions** to turn on actions.
   - Under If, choose to apply actions if the result is a **Pass** or a **Fail**.
   - Under Then, choose all the actions to apply:
     - **Save** a waveform file
     - **Stop** the test
     - Sound an **Alarm**
     - Emit a **Pulse** from the AUX OUT connector. When taking this action, also go to **Utilities > Utilities Setup > Aux Output** and choose to **Use Auxilliary Output For Pass/Fail**.
     - Capture the screen and process it according to your **Hardcopy** (Print) selection
     - Create a **LabNotebook** Entry

   Use the **Clear All** button to clear all the action checkboxes, or **Force Actions Once** to take action once regardless of the test results.

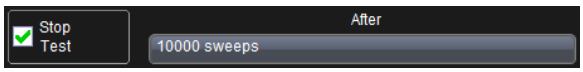
Delay Test

You can delay the start of a test by entering the number of sweeps to wait in **Start Testing After**.

Touch **Clear Sweeps** at any time to reset the test counter.

Stop Test

To stop the test following a specified number of sweeps, rather than a Pass or Fail result, check the **Stop Test** box, then enter the desired number of sweeps in **After**.



Saving Data (File Functions)

This topics in this sections explain the instrument's File menu functions.

Save/Recall features allow you to save and restore setups, waveform data, table data, screen captures, LabNotebooks, and reports. You can use Print or E-mail to share these files.

LabNotebook is Teledyne LeCroy's proprietary tool for capturing a composite file containing waveform data, oscilloscope setups and display, which can be output to a preformatted report. An important feature of LabNotebook is Flashback, which enables you to restore an acquisition (and the setups used to create it) to the oscilloscope screen simply by recalling a LabNotebook entry.

Save

Oscilloscope setups (configurations), waveform data, tabular data, and the display can all be saved in multiple formats.

To save them all as a composite LabNotebook file, see Save LabNotebook.

Save Waveform

The Save Waveform function saves trace data either to internal memory (M1-M*x*) or to a file in text or binary format. The source waveform can be any type of trace; a channel, math function, zoom, or even another memory. Waveform files can be recalled into an internal memory, from where they can be restored to the display.



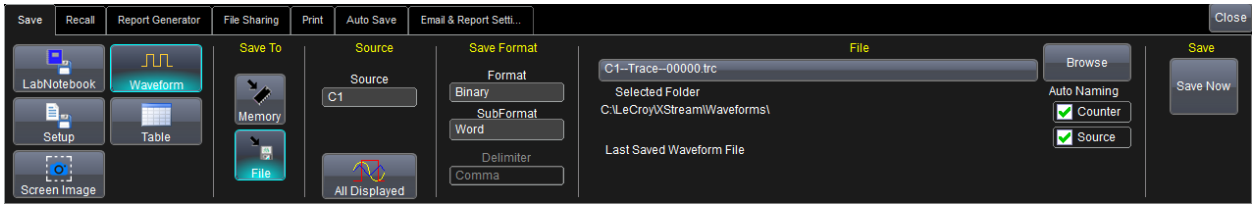
Note: Only files saved in Teledyne LeCroy binary format (.TRC) can be recalled to the display.

Save Waveform To Memory

1. From the menu bar, choose File > Save Waveform.
2. Choose to Save To Memory.
3. Choose the Source trace you are saving.
4. Optionally, select Show On Save to display the memory trace along with its source as soon as it is saved.
5. Touch the Save button directly to the left of the selected Destination memory slot. The date/time stamp of the saved memory will appear at that location.

Save Waveform To File

1. From the menu bar, choose **File > Save Waveform**.
2. Choose to Save To **File**.



3. Choose the **Source** waveform. To quickly save all displayed waveforms separate trace files, touch the **All Displayed** button.
4. Select a file **Format**:
   - **Binary**, 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. Binary files can be converted to ASCII using Teledye LeCroy utilities such as WaveStudio.
   - **ASCII** text file (.txt). Also touch **Delimiter** and select a character from the pop-up menu.
   - **MATLAB** text file (.dat).
   - **Excel** text file (.csv).
   - **MathCad** text file (.prn).
   - **Audio** .wav file.
   - **WaveML**, Teledyne LeCroy's proprietary .xml format used to save persistence maps, eye diagrams, histograms, and digital traces. This option is only active if the source is compatible.
5. Depending on your 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. The system will auto name the file "Trace" prefixed by the source and followed by a counter number. To change the file name or Selected Folder, enter the full path and name in **File**, or touch

Browse and use the File Browser. By default, trace files are saved to the D:\Waveforms folder on the instrument hard drive.

7. If you do not want to use the Source prefix or Counter number, deselect them.
8. Touch Save Now.

Save Setup

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

Save Setup to Memory

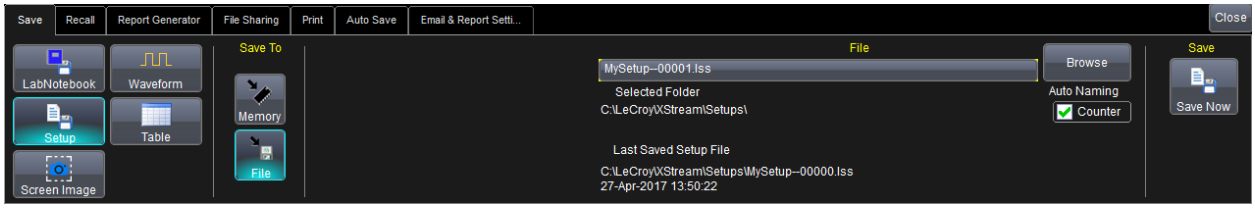
1. From the menu bar, choose File > Save Setup.
2. Choose to Save To Memory.



3. If desired, touch one of the Setup slots and enter a name for the memory. The default name will be Panel*n*. Try to select an empty slot, as anything currently in it will be overwritten.
4. Touch the Save button directly to the left of the selected Setup slot. The save date/time is displayed above the Setup field.

Save Setup to File

1. From the menu bar, choose File > Save Setup.
2. Choose to Save To File.

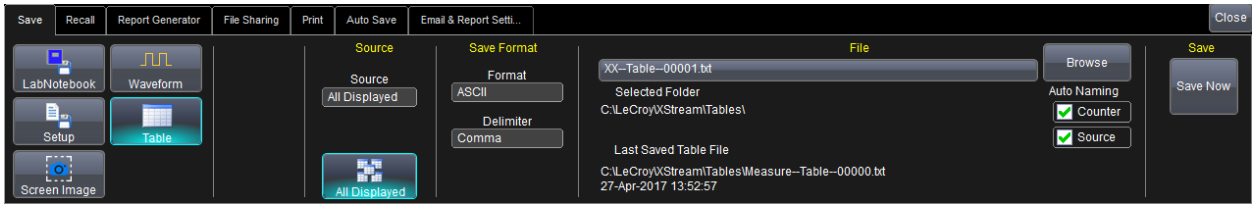


3. The system will auto name the file "DSOSetup" followed by a counter number. To change the file name or Selected Folder, enter the full path and name in File, or touch Browse and use the File Browser. By default, setup files are saved to the D:\Setups folder on the instrument hard drive.
4. If you do not want to use the Counter number, deselect it.
5. Touch Save Now.

Save Table

The Save Table function saves tabular measurement data displayed on screen to an Excel or ASCII file.

1. From the menu bar, choose **File > Save**, then choose **Table**.



2. To save only one of the tables displayed, touch **Source** and navigate to the selection.

   By default, data from all visible tables are saved to separate files. To quickly restore this selection, touch the **All Displayed** button.

3. Choose a format of **ASCII** (.txt) or **Excel** (.csv). If you selected **ASCII** format, also touch **Delimiter** and choose a character.

4. The system will auto name the file "Table" prefixed by the source and followed by a counter number. To change the file name or Selected Folder, enter the full path and name in **File**, or touch **Browse** and use the File Browser. By default, table files are saved to the D:\Tables folder on the instrument hard drive.

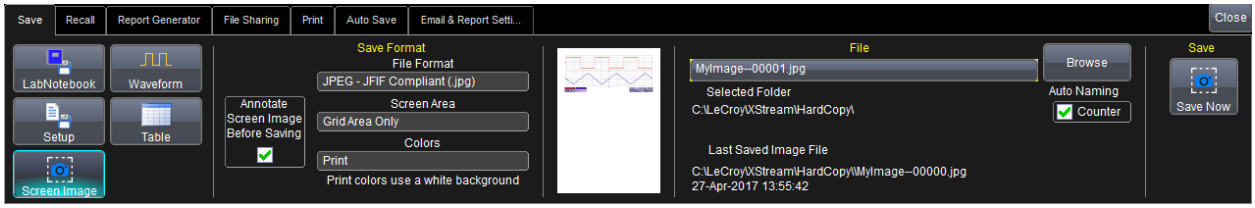
5. If you do not want to use the **Source** or **Counter** number, deselect them.

6. Touch **Save Now**.

Save Screen Image

The full touch screen display or selected portions of it can be captured, saved to an image file, and marked with custom annotations.

1. From the menu bar, choose **File > Save**, then choose **Screen Image**.



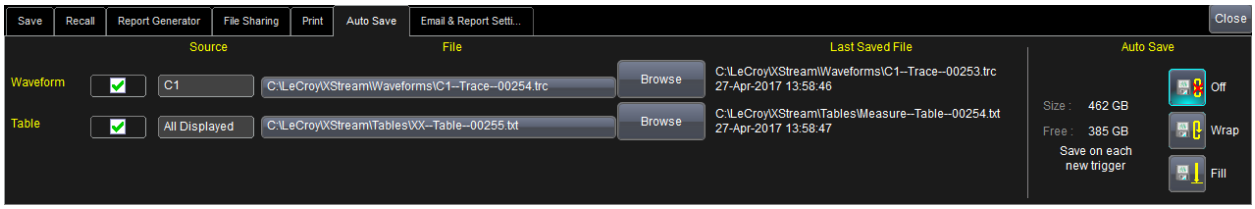
2. Choose a **File Format** of .JPEG, .PNG, .TIF, or .BMP.
3. To mark up the screen image using the drawing tools, select **Annotate Screen Image Before Saving**.
4. Select the **Screen Area** captured:
   - Grid Area Only includes any visible waveform grids, tables, and descriptor boxes.
   - DSO Window includes the above plus any open dialogs, menu bar, and message bar.
   - Full Screen is the full Windows display, including other visible applications and desktop.
5. Choose the **Colors** used in the capture:
   - Standard uses the screen colors on a black background as it appears on the instrument.
   - Print (default) uses your print color palette (set in Preferences) on a white background to save ink.
   - Black & White captures the image in grayscale.
6. The system will auto name the file "LeCroy" followed by a counter number. To change the file name or Selected Folder, enter the full path and name in **File**, or touch **Browse** and use the File Browser. By default, image files are saved to the D:\Hardcopy folder on the instrument hard drive.
7. Touch **Save Now**.
8. If you've turned on annotations, use the Drawing toolbar to mark up the screen image. Click **Done** when finished.

Auto Save

Data that appears on the oscilloscope display, such as waveforms, measurement readouts and decoder data, can be very dynamic and difficult to read from the oscilloscope unless you stop the acquisition.

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

1. Choose **File > Save** and open the **Auto Save** dialog.



2. Select to save **Waveforms** and/or **Tables**.
3. Select the **Source** from which to save.
4. By default, waveform files are stored in D:\Waveforms, and tables are stored in D:\Tables on the instrument hard drive. Optionally:
   - Enter a different path in **File**, or **Browse** to and select the path.
   - Enter a more descriptive root file name than "Trace" or "Table." Auto Saved files will always be named using the Source prefix and a Counter number suffix around the root.
5. Choose one of the **Auto Save** options: **Wrap** (old files overwritten when buffer filled) or **Fill** (no files overwritten, excess is truncated).

   If you have frequent triggers, it is possible you will eventually run out of storage space. Choose Wrap only if you're not concerned about files persisting on the instrument. If you choose Fill, plan to periodically delete or move files off the instrument, or you will lose new data.
6. To enable these selections, choose **Configure Auto Save** from the pop-up dialog that appears.
7. As soon as there is a trigger, files will begin to be saved. Choose **Disable Auto Save** to stop saving files while the acquisition is in process.

   You can also choose **Off** on the Auto Save dialog to disable this feature in between acquisitions.

Recall

Setups saved to internal memory or to .LSS file can be recalled to restore the oscilloscope to the saved state.

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

To recall and modify saved LabNotebook files, see Recall LabNotebook.

Recall Waveform

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

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



2. Touch **Browse** and navigate to the file.

   If the file is in the Selected Folder, you can use the **Up /Down Arrows** to cycle through the available files until the desired file is selected.

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

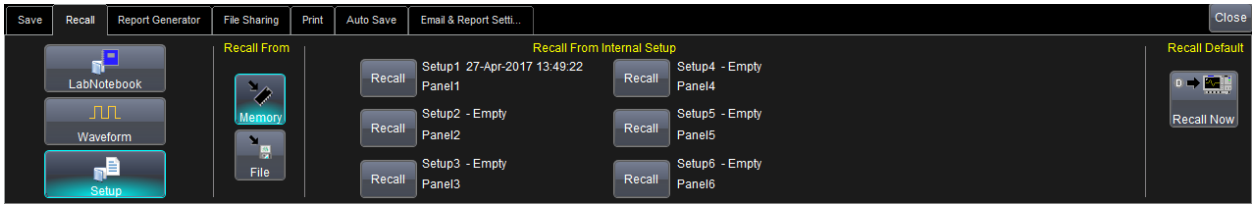
3. Select a **Destination** memory slot into which to recall the file.
4. Mark **Show on Recall** to display the trace on the grid.
5. Touch **Recall Now**.

Recall Setup

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

Recall Setup from Memory

1. Choose to Recall From **Memory**.



2. Touch one of the six **Recall** buttons under **Recall From Internal Setup...**.

Note: If a setup has been stored to a location, it is labeled with the save date/time. Otherwise, the slot is labeled **Empty**.

Recall Setup from File

1. Choose to Recall From **File**.



2. Enter the path to the **File**, or touch **Browse** and navigate to the desired file.

   If the file is in the Selected Folder, you can use the **Up /Down Arrows** to cycle through the available files until the desired file is selected.

3. Under Recall File, touch **Recall Now**.

Recall Default Setup

Restore the factory default settings by pressing the front panel **Scope Setup** button and choosing **Default Setup**.

You can also restore default settings by choosing **File > Recall Setup** from the menu bar and selecting **Recall Default**.

LabNotebook

The LabNotebook feature allows you to create and save composite files containing a screen capture of all displayed waveforms, as well as all waveform and setup data at the time of capture.

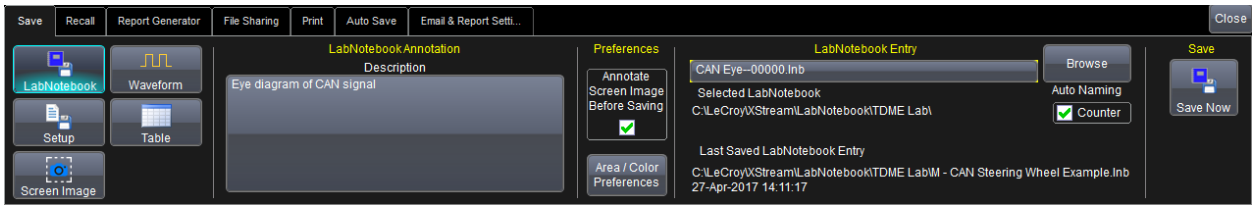
LabNotebooks can be output to a preformatted .PDF, .RTF, or .HTML report. You can also upload your own report layout if you prefer not to use the default.

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

Save LabNotebook

All LabNotebook files (.LNB) are composed of a screen image (.PNG), a setup file (LSS), and a waveform file (.TRC) for each waveform displayed. When creating LabNotebooks, you choose how to handle the screen image component, which is the basis for the Report Generator output, and to which you can add a description and other hand-drawn annotations.

1. Choose **File > Save LabNotebook** from the menu bar.



2. Optionally, enter a **Description**. This text appears whenever the file is recalled and on reports.
3. To mark up the screen image using the drawing tools, select **Annotate Screen Image Before Saving**.

   To skip this step, deselect the checkbox. You can recall the LabNotebook later to add a description and annotations.
4. To change the area of the screen captured and the colors used, touch **Area/Color Preferences** and make your selections on the pop-up:

   - Use Print Colors (default) uses your print color palette (set in Preferences) on a white background to save ink. Deselect this to capture the display using the screen colors on a black background as it appears on the instrument.
   - Grid Area Only includes any visible waveform grids, tables, and descriptor boxes.
   - DSO Window includes the above plus any open dialogs, menu bar, and message bar.
   - Full Screen is the full Windows display, including other visible applications and wallpaper.

   Touch **Close** to save your changes.
5. The system will auto name the file "MyLabNotebook" followed by a counter number. To change this to something more descriptive, or to change the file path, enter it in **LabNotebook Entry**, or touch

Browse and use the File Browser to make a selection. By default, LabNotebook files are saved to the D:\LabNotebook folder on the instrument hard drive.

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

6. If you do not want to use autonumbering to identify files, deselect Counter.
7. Touch Save Now.
8. If you've left on annotations, use the Drawing toolbar to mark up the screen image. Click Done when finished.

LabNotebook Drawing Toolbar

The basic LabNotebook is a screen capture of the display as it was at the time of entry, along with the setup and waveform data. If you have selected to Annotate Screen Image Before Saving, the capture is immediately displayed in the Drawing window. You can later add Annotations by recalling the LabNotebook into the Preview window and choosing to Annotate Screen Image.

Markup tools are available from the toolbar along the top of the window.



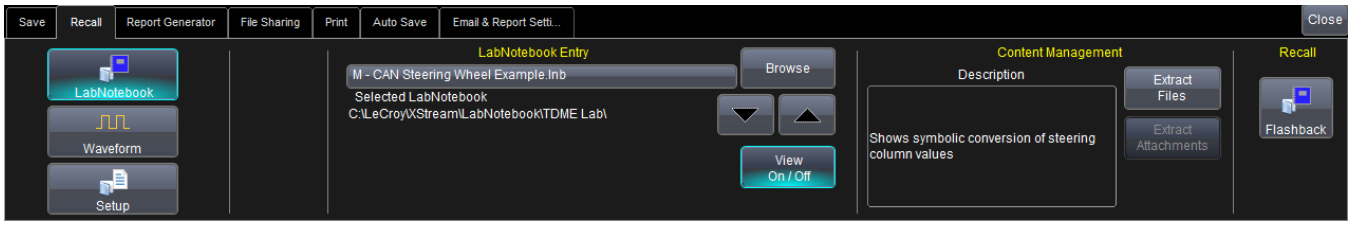
To use any tool, touch the icon, then touch on the image where you wish to draw or add text. From left to right, the tools are:

Tool	Function
Pencil	Draw in freehand. Maintain contact with the screen to make a continuous mark. Once you release, you can touch-and-drag the object to any point on the image.
Circle	Draw a circle around a waveform feature that you want to emphasize. Touch-and-drag across the diameter of the circle. When you release, the circle is placed. You can drag the circle to any location on the image.
Arrow	Draw lines with arrowheads for placing callouts. You can rotate these lines through 360 degrees or drag them to any location on the image.
Text	Open a textbox for placing labels/annotations on the image. Touch the point on the image to place the label, then enter the text in the pop-up dialog. Once placed, you can resize the textbox or drag it to any location on the image.
RGB Selectors	Quickly change the line color. Just touch the color icon, then choose the next drawing tool.
More	Activates a Custom line color field. The default color is Yellow. To choose another, touch the color swatch, then select from the Color dialog. You can enter RGB values, or choose from the spectrum. After saving, the new color appears in the Custom field. This remains the markup color until you choose another.
Erase Selected / Erase All	Remove selected drawing objects. Erase All will also undo any Custom color selection.
Undo	Cancel the last action. Use it to restore any objects you inadvertently erased.
Move	Undock the drawing toolbar so you can move it anywhere on the display. This helps to keep tools handy when working on a particular area of a waveform. Touch the button again to restore the toolbar to the top of the Drawing window.
Done	Save the annotations with the image and close the Drawing window.

Recall LabNotebook

Once a LabNotebook Entry is saved, Recall it to view and modify descriptive information or manage attachments and component files. Flashback to restore the oscilloscope to the state saved in the LabNotebook file.

Choose File > Recall LabNotebook from the menu bar.



Flashback

Flashback restores the waveforms and setups saved with the LabNotebook Entry, so you can later analyze the inputs that resulted in that capture. Some result data *not* included in Flashback are:

- Persistence data, although it is saved with the .LNB file and appears on reports.
- Histogram data over 16-bits. Histograms internally have a 32-bit resolution, but when stored into a trace file and recalled during Flashback, they are clipped to 16-bits.
- Floating point waveforms resulting from certain math operations that have much higher resolution than 16-bits. This extra resolution is not preserved when traces are recalled using Flashback.
- Cumulative Measurements in process when Flashback is entered. When Flashback is used, they lose their history and show instead only the results from the stored waveforms, not including any data taken from interim acquisitions.

1. In LabNotebook Entry, enter the path to the LabNotebook, or touch Browse and navigate to the file.
2. Touch the Flashback button.
3. To exit Flashback, touch the Undo button at the far right of the menu bar.

Recall and Preview

1. Enter the path to the file in **LabNotebook Entry**, or touch **Browse** and navigate to the file.
2. Touch **View On/Off** to open it in the Preview window.
3. While the file is in preview, you may Edit Description, Annotate Screen Image, or Manage Attachments.
4. Optionally, choose to **Save**, **Print**, or **Email** the LabNotebook in the output report format shown in the preview. Select **Close** to close the Preview window.

Edit Description

You can modify the description and annotations saved with the LabNotebook Entry. Setups and waveform data originally saved with the LabNotebook cannot be changed.

1. Browse to and select the **LabNotebook Entry**. If the file is in the Selected Folder, just use the Up/Down Arrow keys to select it.
2. Select **View On/Off** to preview the file.
3. From the top of the preview window, choose **Edit Description**.
4. Touch the pop-up description to open the virtual keyboard and edit the text.
5. Touch **OK** twice to save changes.

Annotate Screen Image

1. Browse to and select the **LabNotebook Entry**. If the file is in the Selected Folder, just use the Up/Down Arrow keys to select it.
2. Select **View On/Off** to preview the file.
3. From the top of the preview window, choose **Annotate Screen Image** and use the drawing tools.
4. Click **Done** when finished to close the Drawing window and return to the preview.

Manage Attachments

To append external files to the LabNotebook composite (such as images of the DUT, mask files, test scripts, or anything relevant to the entry):

1. Browse to and select the **LabNotebook Entry**. If the file is in the Selected Folder, just use the Up/Down Arrow keys to select it.
2. Select **View On/Off** to preview the file.
3. From the top of the preview window, choose **Manage Attachments** and select the files to attach. The file list will build in the lower part of the File Attachments browser.
4. Click **Update Attachments** to finish.

To later remove appended files, Manage Attachments and deselect the files from the list. Update Attachments again to refresh the list.

Convert File Format

Individual entries in legacy .ZIP format Notebooks can be converted to the new LabNotebook file format.

1. In LabNotebook Entry, **Browse** to and select the legacy .zip file from the left side navigation pane of the File Browser.

   Note: Be sure to select the .zip file from the left pane so that its sub-entries appear in the right pane and the Save buttons appear on the browser. The file should appear in Current Path on the file browser.

2. To batch convert all entries into separate LabNotebook files, at the top of the file browser choose **Save All As LNB Files**.

   To convert a single entry, select it from the **Name** list (right side), then choose **Save As LNB File**.

New LabNotebooks of the same name as the original entries are created in the D:\XPort folder. These can be selected the same as any other .LNB files for Flashback, editing, reports, or extraction.

Extract Files

The component files that make up the LabNotebook composite (.PNG, .LSS, and .TRC) and any appended files can be extracted into separate files.

1. Enter the path to the **LabNotebook Entry**, or **Browse** to the file.
2. To extract all files, under Content Management, choose **Extract Files**.

   To extract only the attachments, choose **Extract Attachments**.

3. Navigate to the folder where you want the files extracted, and choose **Extract Now**.

   A folder of the same name as the original LabNotebook containing the separate files will be created at that location.

Report Generator

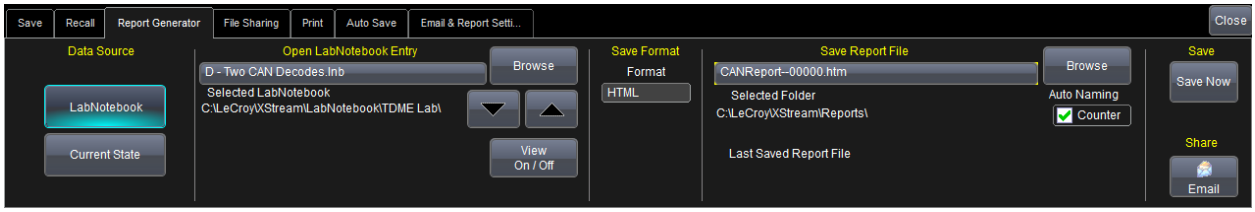
The Report Generator feature allows you to output a preformatted report consisting of an annotated screen image and a summary of the setups in place when it was captured. The report can be sourced from an existing LabNotebook, or it can be newly generated from the current state of the oscilloscope.

The logo and output template used to create the report can be changed on the Email & Report Settings dialog.

From LabNotebook

This procedure outputs an existing source as is. If you want to further annotate or change the description of a LabNotebook file, first recall the LabNotebook.

1. Choose **File > Report Generator** from the menu bar.



2. **Browse** to and select the **LabNotebook Entry**. If it is in the Selected Folder, just use the Up/Down Arrow keys to select it.
3. Optionally, use **View On/Off** to preview the selected file.
4. Choose a **Save Format** of HTML, .RTF, or .PDF.

> **Note:** Only HTML reports can later be saved to .RTF or .PDF format. The .RTF and .PDF reports can only be combined into another report of the same format.

5. By default, the system will auto name the file "Report" followed by a counter number. To change the file name or the Selected Folder, enter the full path and name in **Save Report File**, or touch **Browse** and use the File Browser. By default, report files are saved to the D:\Xport folder on the instrument hard drive.

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

6. If you do not want to use autonumbering, deselect **Counter**.
7. To save the report, choose **Save Now**.

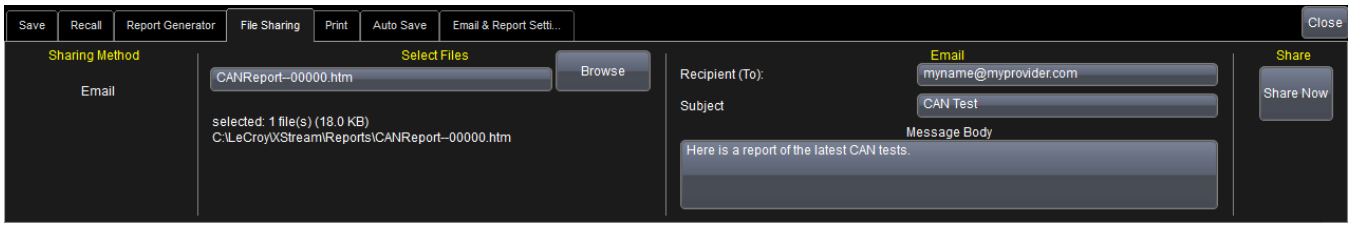
From Current State

If you are creating a report from the current state of the instrument, the procedure is much like that above, except that you will also be able to enter a new **Description**, **Annotate Screen Image**, and select your capture **Area/Color Preferences** as you would normally do when first creating a LabNotebook. See Save LabNotebook for an explanation of these steps.

However, when using the Report Generator from Current State, you will not have saved a composite .LNB file that can later be recalled or extracted into its component files. You will only have the preformatted report file.

Share

Use the File Sharing dialog to email files from the instrument. Configure the oscilloscope for email on the Email & Report Settings dialog.



1. Choose **File > File Sharing** from the menu bar.
2. Use the File Browser to select all the files to be emailed.

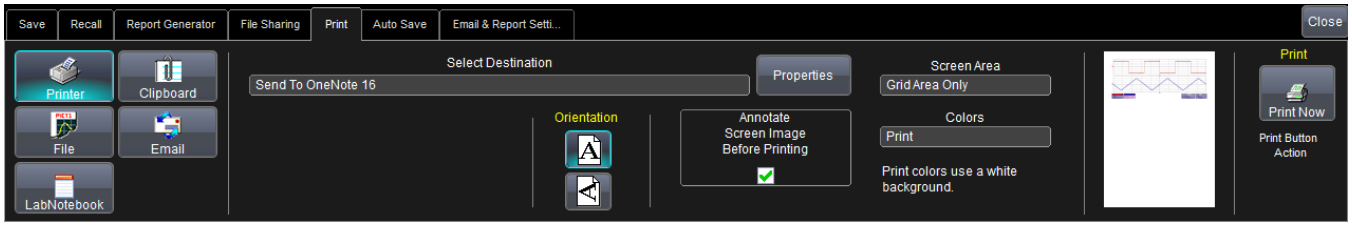
   Use CTRL + Click to select multiple files.
3. Optionally, change the email **Recipient(s)**. This field defaults to whatever is in your Email Settings.
4. Enter a **Subject** line and **Message Body**, then **Share Now**.

Print

Print dialog settings control how the front panel **Print** button behaves. Print captures an image of the touch screen display, but there are several options as to what it does next with the image:

- Send it to a **Printer**
- Copy it to the Windows **Clipboard**
- Save it to an image **File** using your current Screen Image Preferences
- **Email** an image file using your current Screen Image Preferences and Email Preferences
- Create a new **LabNotebook** file using your current LabNotebook Preferences

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



Printer Settings

If sending to a printer:

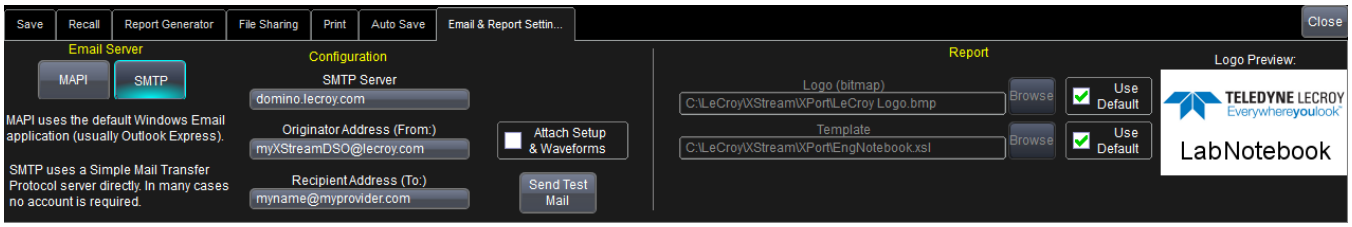
1. Touch **Select Destination** and choose a connected printer. Touch **Properties** to change any of the printer properties using the standard Windows Print dialog.
2. Choose a paper orientation of **Portrait** or **Landscape**.
3. Select **Annotate Before Saving** to mark up captures before they print. Each time you press Print, the capture will open in the Drawing Tools window. When you are **Done** drawing, the image prints.

Capture Settings

1. Select a **Screen Area**:
   - Grid Area Only includes any visible waveform grids, tables, and descriptor boxes.
   - DSO Window includes the above plus any open dialogs, menu bar, and message bar.
   - Full Screen is the full Windows display, including other visible applications and wallpaper.
2. Choose the **Colors** used in the capture:
   - Standard uses the screen colors on a black background as it appears on the instrument.
   - Print uses your print color palette (set in Utilities > Preference Setup) on a white background
   - Black & White captures the image in grayscale.

Email & Report Settings

This dialog contains additional settings used by the File Sharing and Report Generator functions.



Email Settings

1. Select to use MAPI or SMTP.
2. If you chose SMTP, touch SMTP Server and enter the network address of your mail server.
3. Touch Originator Address (From:) and enter the instrument's e-mail address.
4. Touch Default Recipient Address (To:) and enter the recipient's e-mail address.
5. Optionally, select to always Attach Setup & Waveforms associated with image files selected for email.
6. Use Send Test Mail to send a confirmation message to ensure proper e-mail configuration.

Report Settings

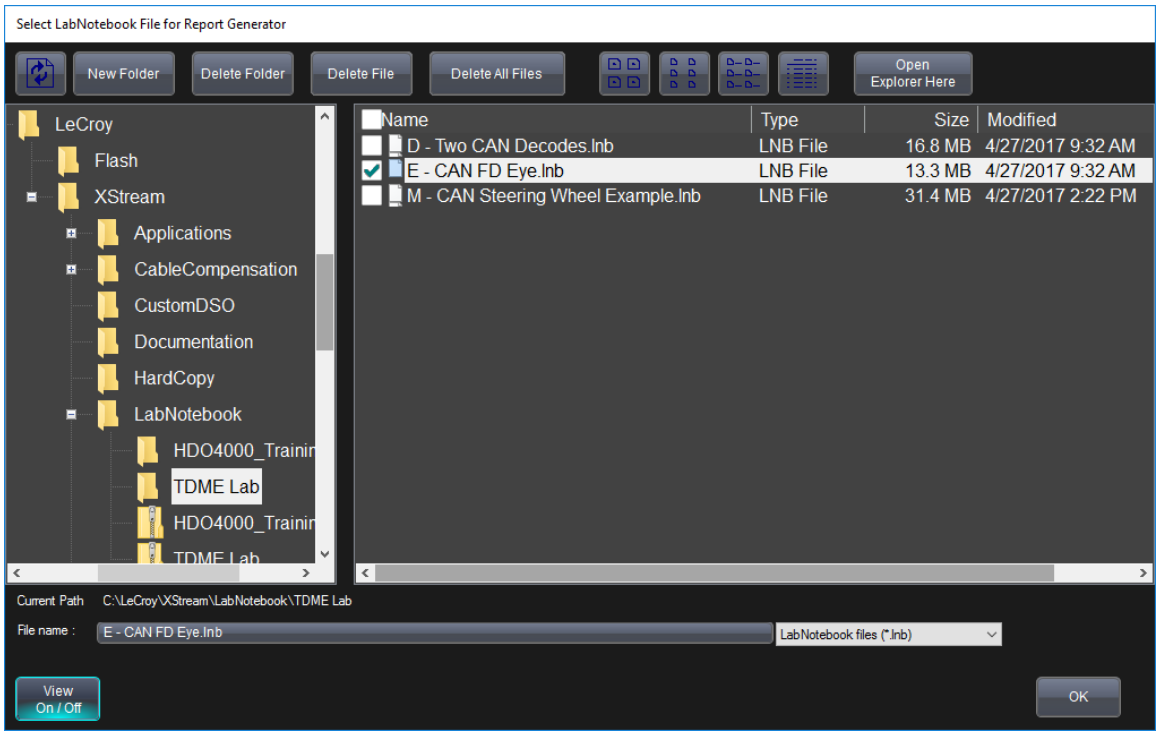
The default report template uses our logo as a placeholder. You can replace this with your logo. Logo files should be in bitmap (.bmp) format and not exceed 100 pixels high by 180 pixels wide. Place the file in the D:\Xport folder.

You can also use your own report template. Templates must be saved as .xsl or .xslt files and placed in the D:\Xport folder.

Deselect Use Defualt next to each item you want changed, then Browse to and select the new files.

Using the File Browser

Selecting **Browse** on any of the File menu dialogs opens a File Browser that enables you to perform basic disk utility functions, as well as making file path/name and format selections. The additional file actions available depend on the actions available on the underlying dialog (Save file, Email report, etc.).



Disk Utilities

New Folder, **Delete Folder**, **Delete File**, and **Delete All Files** can be used to change your file system prior to saving new files. Be sure to first select the folder from the navigator or file from the file list.

File Path and Name

The **Current Path** is shown immediately below the navigator pane. To change it, use the navigator to select a different folder.

To select an existing file, use the file list to the right of the navigator. The buttons immediately above the file list let you change how items appear: icons only, details, etc. If you do not see the file you seek, try using the drop-down next to **File name** to change the file type to **"All files (*.*)"**.

Most file types have a default root file name (e.g., "MyLabNotebook"), to which an autogenerated prefix and/or suffix is added. To change this, enter the new **File name**. If you are autogenerating, the root file name for a given file type persists until changed.

Auto Naming Selections

The checkboxes to turn on/off the Source prefix and the Counter number suffix used to autogenerate file names will appear on the File Browser. These selections are linked to those on the underlying dialog, and changing the value in either place causes it to change everywhere.

Note: If you change any part of the file path/name or format on the File Browser or the dialog, the Counter number will reset to the last number in the sequence associated with that value.

Example: Changing "Decode1-Table-0002" to "Decode1-CANFDTable-..." will reset the counter to "Decode1-CANFDTable-0000" if there is no prior file named "Decode1-CANFDTable-..." .

Actions

The Close button accepts the selections you made on the File Browser and closes the browser window. If you do not need to make further entries on the dialog to complete your task, you can choose to Save, Recall, Flashback, Email, etc. right from the File Browser.

Utilities

Utilities settings primarily control the instrument's interaction with other devices/systems.

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

Disk Utilities enable you to perform commonplace Windows file functions.

Utilities Dialog

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



Date/Time Setup and System Status buttons open their corresponding dialogs.

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

Show Windows Desktop minimizes the X-Stream application window. Maximize the application by touching the display icon located at the lower-right of the desktop.

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

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

Status

The Utilities Status dialog displays information about your instrument including model number, serial number, firmware version, and installed hardware and software options.

Choose **Utilities > Utilities Setup** from the menu bar, then touch the **Status tab**.

Or

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

Remote Control

The Remote dialog contains settings to configure remote control of the instrument and also network access. Supported remote control protocols are:

- TCPIP (Ethernet). If you choose this option, also install Teledyne LeCroy's VICP drivers on the controller. These are included in the VICP Passport plug-in, available free from teledynelecroy.com/support/softwaredownload under Oscilloscope Downloads > Software Utilities. The instrument uses Dynamic Host Configuration Protocol (DHCP) as its default addressing protocol. You can assign a static IP address using the standard Windows network setup menus.
- LXI (Ethernet)
- GPIB and LSIB. These selections are active only if the instrument has the GPIB or LSIB hardware option installed. Connect the controller to the respective port.

Note: Full remote control setup requires installation and configuration of software on the controller, as well. See the *MAUI Oscilloscopes Remote Control and Automation Manual* for full instructions on making the remote connection and sending remote commands. The steps shown here are only those performed on the oscilloscope to enable the connection.



Set Up Remote Control

1. From the menu bar, choose Utilities > Utilities Setup, then touch the Remote tab.
2. On the Remote dialog, make a Control From selection.
3. If using TCPIP and wish to restrict control to specific network clients, choose Yes under Security. Enter the IP addresses or DNS names of the authorized controllers in a comma-delimited list.

Configure the Remote Control Assistant Event Log

The Remote Control Assistant monitors communication between the controller and instrument. 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

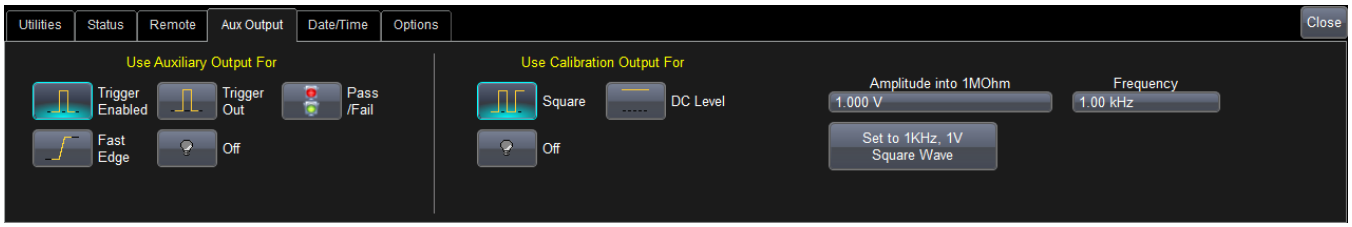
1. From the menu bar, choose Utilities > Utilities Setup, then touch the Remote tab.
2. Touch the Show Remote Control Log button. The Event Logs pop-up is shown.
3. Enter a log file name in DestFilename, or touch Browse and navigate to an existing file.

   Note: New contents will overwrite the existing content; it is not appended.

4. Touch Export to Text File.

Auxiliary Output

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



Auxiliary Output

Trigger Enabled sends a pulse when the trigger is ready (Ready indicator lit), but not necessarily triggered. It can be used as a gating function to trigger another instrument. Enter the desired pulse Amplitude.

Trigger Out sends a pulse upon a trigger (TRIG'D Indicator lit). Enter the desired pulse Amplitude.

Pass/Fail generates a pulse when Pass/Fail testing is active and conditions are met. Enter the desired Pulse Duration.

Reference DC Level. Enter the desired Amplitude.

Off disables auxiliary input/output.

Calibration Output

Choose the signal to be output from the Cal Out hook on the front of the instrument:

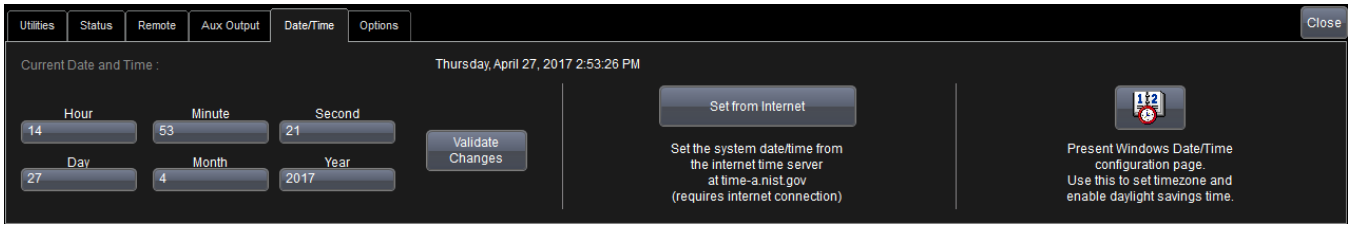
- Square wave signal. Enter the wave Frequency and Amplitude into 1 MΩ, or choose to Set to 1 kHz, 1 V Square Wave.
- Reference DC Level. Enter an Amplitude into 1 MΩ.

Off disables calibration output.

Date/Time

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

To access the Date/Time dialog, choose **Utilities > Utilities Setup** from the menu bar, then touch the **Date/Time tab**.



Manual Method

Enter the **Hour**, **Minute**, **Second**, **Day**, **Month**, and **Year**, then touch the **Validate Changes** button.

Internet Method

This method uses the Simple Network Time Protocol (SNTP) to read the time from time-a.nist.gov. The instrument must be connected to an internet access device through a LAN (Ethernet) port.

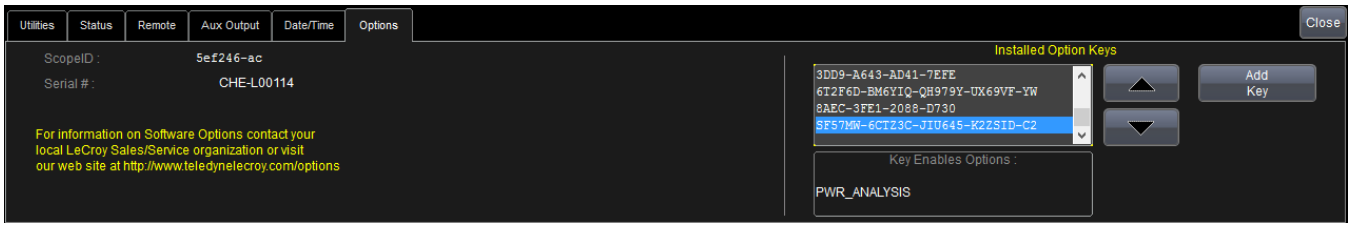
If your connection is active, touch the **Set from Internet** button.

Windows Method

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

Options

Many optional software packages are available to extend the Analysis functions of the instrument. When you purchase an option, you will receive a key code by email that activates the new functionality. Use the Options dialog to activate software options by installing the key code. This dialog also displays the ScopeID and Serial #.



To install a key:

1. From the menu bar, choose Utillities > Utilities Setup, then 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 OK to enter the information.

   The Key Code is added to the list of Installed Option Keys. You can use the Up/Down buttons to scroll the list. The software option that each key activates is displayed below the list.
4. Restart the X-Stream application: choose File > Shutdown, then double-click the Start DSO icon on the desktop.

Disk Utilities

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

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

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



Delete a Single File

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

Delete All Files in a Folder

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

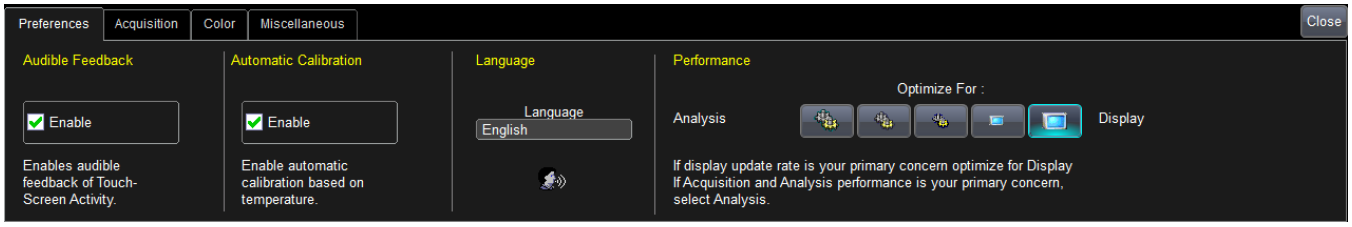
Create a New Folder

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

Preferences Settings

Preference settings have mostly to do with the appearance and performance of the instrument itself, rather than its interaction with other devices/systems. These setting are also called "non-volatile," because they are not lost and do not change when the oscilloscope is restarted or a setup panel is recalled.

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



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 automatic calibration routine. When enabled, the instrument re-calibrates whenever there is a change in acquisition settings that affects the ADC configuration.

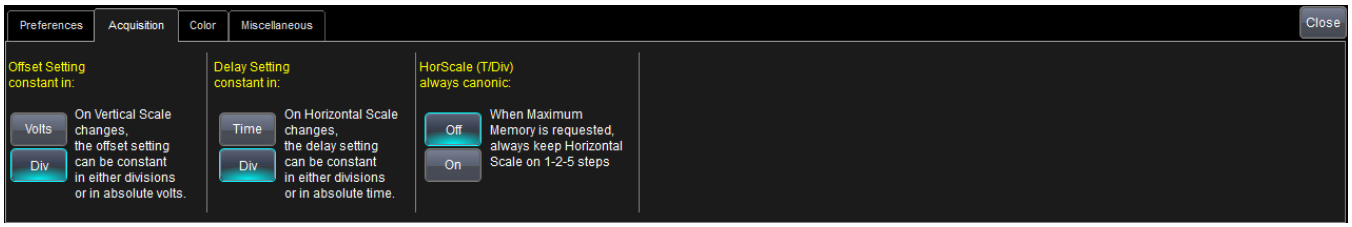
Language sets the language used on the touch screen display.

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

There are also tabs linking to Acquisition, Color, and Miscellaneous settings.

Acquisition

The Acquisition preference settings determine how traces behave as Vertical Offset or Horizontal Delay changes. To access them, choose **Utilities > Preference Setup** and open the **Acquisition** dialog.



Offset Setting constant in:

- **Volts** 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 Delay indicator relative to the 0 point at the center of the grid.
- **Div**(isions) keeps the Horizontal Delay indicator stationary. The trigger point remains on the grid as you increase the timebase; whereas, if Time is selected, the trigger point could move off the grid.

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

Color

Color dialog settings assign the colors used for channel, math, and memory traces. All dialogs, tables, and trace descriptor boxes will match the color of the trace assigned here. You can choose different colors to be used on the instrument and in print.

For convenience, you can Preview print colors to see how the settings will appear in print output.

Note: Print colors are used only when the Colors control is set to Print on any of the File menu dialogs (Save Screen Image, Print, etc.). Otherwise, the Screen colors are used for print output as well as on screen.

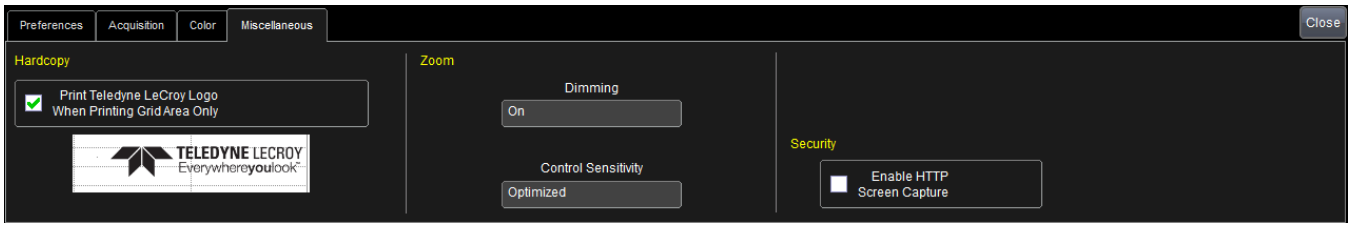


To make any setting, just touch the color swatch for either Screen or Print next to the trace number, and make a selection from the Color pop-up menu.

Touch Factory default colors to recall the original color settings for your instrument.

Miscellaneous

These other Preference settings are located on the Miscellaneous dialog.



To add the Teledyne LeCroy logo to print output, check Print Teledyne LeCroy Logo When Printing Grid Area Only. This identifies the instrument as the source of the image.

Dimming darkens/shades those areas of the source waveform that are not part of the Zoom trace.

Control Sensitivity adjusts the sensitivity of the front panel knobs. Optimized applies an acceleration algorithm to the knobs. Legacy detects rotation of the front panel knobs in a manner similar to our legacy oscilloscopes.

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

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

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

Maintenance

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

Touch Screen Calibration

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

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

Restart/Reboot Instrument

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



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

To reboot the instrument, which includes restarting the OS:

1. From within the oscilloscope application, choose **File > Shutdown** from the menu bar. From the Windows desktop, use the Windows Start Menu Shutdown command.
2. Wait 10 seconds, then press the **Power button** on the front of the instrument.

Firmware Update

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

The firmware update procedure *does not* modify or delete any saved panel setups, waveforms, screen captures, calibration constants, or other data stored on the D: drive.



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

1. Exit the X-Stream 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 **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 instrument 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 instrument.
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.
   - Touch Screen Driver - required to use the display as a touch screen (you can opt to use a mouse).
   - LSIB Package - required for LSIB hostboard or hostcard.
   - SPARQ Package - required to drive a connected SPARQ from the instrument.

   Click **Install** when done.
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** *after* you see that programming has completed to return to the X-Stream Setup wizard.
10. When installation is complete, choose **Reboot now** and click **Finish**.

Technical Support

Live Support

Registered users can contact their local Teledyne LeCroy service center at the number listed on our website.

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

teledynelecroy.com/support/techhelp

Resources

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

teledynelecroy.com/support/techlib

The Datasheet published on the product page contains the detailed product specifications.

Service Centers

For a complete list of offices by country, including our sales & distribution partners, visit:

teledynelecroy.com/support/contact

Teledyne LeCroy
700 Chestnut Ridge Road
Chestnut Ridge, NY, 10977, USA
teledynelecroy.com

Sales and Service:
Ph: 800-553-2769 / 845-425-2000
FAX: 845-578-5985
contact.corp@teledynelecroy.com

Support:
Ph: 800-553-2769
customersupport@teledynelecroy.com

Returning a Product for Service

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

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

1. Remove all accessories from the 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.

Certifications

Teledyne LeCroy certifies compliance to the following standards as of the time of publication. See the *EC Declaration of Conformity* document shipped with your product for the current certifications.

EMC Compliance

EC Declaration of Conformity- EMC

The instrument meets the intent of EC Directive 2014/30/EU for Electromagnetic Compatibility. Compliance was demonstrated to the following specifications listed in the Official Journal of the European Communities:

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

Electromagnetic Emissions:

EN 55011:2010, Radiated and Conducted Emissions Group 1, Class A [2 3]

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

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

Electromagnetic Immunity:

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

EN 61000-4-3/A2:2010 RF Radiated Electromagnetic Field, 3 V/m, 80-1000 MHz; 3 V/m, 1400 MHz - 2 GHz; 1 V/m, 2 GHz - 2.7 GHz

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

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

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

EN 61000-4-11:2004 Mains Dips and Interruptions, 0%/1 cycle, 70%/25 cycles, 0%/250 cycles [4 5]

1. To ensure compliance with all applicable EMC standards, use high quality shielded interface cables.
2. Emissions which exceed the levels required by this standard may occur when the instrument is connected to a test object.
3. This product is intended for use in nonresidential areas only. Use in residential areas may cause electromagnetic interference.
4. Meets Performance Criteria “B” limits of the respective standard: during the disturbance, product undergoes a temporary degradation or loss of function or performance which is self-recoverable.
5. Performance Criteria “C” applied for 70%/25 cycle voltage dips and 0%/250 cycle voltage interruption test levels per EN61000-4-11.

*European Contact:**

Teledyne LeCroy Europe GmbH
Im Breitspiel 11c
D-69126 Heidelberg
Germany
Tel: + 49 6221 82700

Australia & New Zealand Declaration of Conformity– EMC

The instrument complies with the EMC provision of the Radio Communications Act per the following standards, in accordance with requirements imposed by Australian Communication and Media Authority:

AS/NZS CISPR 11:2011 Radiated and Conducted Emissions, Group 1, Class A.

AUSTRALIA / NEW ZEALAND CONTACTS:*

RS Components Pty Ltd.
Suite 326 The Parade West
Kent Town, South Australia 5067

RS Components Ltd.
Unit 30 & 31 Warehouse World
761 Great South Road
Penrose, Auckland, New Zealand

*Visit teledynelecroy.com/support/contact for the latest contact information.

Safety Compliance

EC Declaration of Conformity– Low Voltage

The instrument meets intent of EC Directive 2014/35/EU for Product Safety. Compliance was demonstrated to the following specifications as listed in the Official Journal of the European Communities:

EN 61010-1:2010 Safety requirements for electrical equipment for measurement, control, and laboratory use – Part 1: General requirements

EN 61010-2:030:2010 Safety requirements for electrical equipment for measurement, control, and laboratory use – Part 2-030: Particular requirements for testing and measuring circuits

The design has been verified to conform to the following limits put forth by these standards:

- Mains Power Supply Circuits: Overvoltage Category II, instrument intended to be supplied from the building wiring at utilization points (socket outlets and similar).
- Measuring Circuit Terminals: No rated measurement category. Terminals not intended to be directly connected to the mains supply.
- Unit: Pollution Degree 2, operating environment where normally only dry, non-conductive pollution occurs. Temporary conductivity caused by condensation should be expected.

U.S. Nationally Recognized Agency Certification

The instrument has been certified by Underwriters Laboratories (UL) to conform to the following safety standard and bears UL Listing Mark:

UL 61010-1 Third Edition – Safety standard for electrical measuring and test equipment.

Canadian Certification

The instrument has been certified by Underwriters Laboratories (UL) to conform to the following safety standard and bears cUL Listing Mark:

CAN/CSA-C22.2 No. 61010-1-12. Safety requirements for electrical equipment for measurement, control and laboratory use.

Environmental Compliance

End-of-Life Handling



The instrument is marked with this symbol to indicate that it complies with the applicable European Union requirements to Directives 2012/19/EU and 2013/56/EU on Waste Electrical and Electronic Equipment (WEEE) and Batteries.

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

ISO Certification

Manufactured under an ISO 9000 Registered Quality Management System.

Warranty

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

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

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

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

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

Intellectual Property

All patents pertaining to the WaveRunner 6 Zi/HRO can be found on our website at:

teledynelecroy.com/patents/

Windows License Agreement

The oscilloscope application software runs on a Windows® operating system. Teledyne LeCroy's agreement with Microsoft® prohibits users from installing third-party software that is not relevant to measuring, analyzing, or documenting waveforms.

Index

A
acquisition
optimiztion 160
pre-processing 24, 42
sampling mode 48
settings 161
action toolbar 26, 91
activating traces 30
altitude 2
analog inputs 12
attenuation 12
auto naming 152
auto save 138
auto zero 43
AUX connectors 17, 156
averaging 24, 42, 106, 108
B
bandwidth limiting 24, 41
C
cable de-embedding 42
calibration 2, 15, 156, 160, 168
Cascaded 64
channel 40
descriptor box 24
frequency 12
setup 41
cleaning 2
clock
external 54
reference 54
sample 54
color 162
compliance 169
cooling 2
copy function 107
coupling 12, 24, 41
cursor 77-78
controls 79
readout 25, 77
custom measurements 83
D
date and time 157
de-embedding 12, 42
default setup 140
degauss 43
delay 48, 161
post-trigger 48
pre-trigger 48
descriptor box 21, 24, 30
deskew 24, 42
dialogs 26
differential time measurements 102
display 16, 71
controls 72, 163
extended 16
grid 22, 72
optimization 160
persistence 74

E
EC compliance 169
email 148-149
EMC 169
enhanced resolution 42, 108
extended display 16
external monitor 16
F
FFT 111
file browser 151
file structure 151, 159
filtering
bandwidth 41
LabNotebook entries 151
measurements 84
noise 42, 108
firmware 15
update 166
version 153
Flashback Recall 141, 144
frequency 108
response 12, 111
front panel 163
G
gain 41, 161
gating measurements 84
GPIB 154
graphing measurements 91
grid 22, 55
auto grid 71
intensity 72
style 72
H
hard drive 159
histograms 82, 91, 116
WaveScan 126
holdoff 65
horizontal
controls 48
humidity 2
I
import
.trc files 139
setup panels 140
inputs
analog 12
intensity
grid 72
interfaces 12
interleaving 49-50
interpolation 42, 113
inversion 24, 42
IP address 16, 154
L
labelling traces 33, 142
LabNotebook 21, 141-142
recall 144
save 141, 149
language selection 19

logs
remote control event 155
LSIB 154
M
markers 82
mask testing 128
math 103, 105
descriptor box 24
function setup 103
graphing 104
on parameters 86
operators 103
measurements 80, 100-102
cursors 77
custom 83
filtering 84
gating 84
graphing 91, 94-95
histicons 82
list of 96
markers 82
pass/fail testing 128, 130
processing web 88
readouts 80
statistics 81
measurements; status 80
memory 120-121, 133, 135
descriptor box 24
memory length 49
multi-stage triggers 63
multi-zoom 34
O
offset 161
operating environment 2
optimization 160
options 15, 158
P
parameter compare 128, 130
parameter math 86
pass/fail testing 128, 130-131
persistence 74
histogram 116
WaveScan 126
position
trace 34
post-trigger delay 48
power
standby 15
pre-processing 24
pre-trigger delay 48
preferences 160
printing 16, 38, 149, 163
probes 12
settings 43
tip selection 43
processing web editor 88
Q
qualified measurements 84
qualified trigger 63

R
real-time sampling mode 50
recall
LabNotebooks 144
reference clock 17, 54
remote control 16, 154, 163
reports 141, 147
rescaling 42, 114
restart/reboot 165
restore
waveforms 121
returns 168
RH 2
RIS sampling mode 50
rise and fall time 101
roll sampling mode 50
S
safety 1, 170
sample clock 16, 54
sample points 49, 113
sample rate 49-50
sampling mode 48
real-time 50
RIS 50
roll 50
sequence 51
save
data 141
setup panels 141
waveforms 120
screen capture 38, 137, 149
search 122
sequence sampling mode 51, 73
serial triggers 163
service 167-168
setup panels 135, 140-141
sin x 24, 42
software assisted trigger 68
software options 15, 158
sound 160
sparse function 113
support 167-168
system
hibernate 15
on/off 15
status 153
timestamp 157
T
tables 136
auto save 138
technical support 167-168
temperature 2
time parameters 101
timebase 48, 161
clock 54
controls 48
descriptor box 25
top and base 100

touch screen 16, 21
  calibration 165
  language 19
traces
  3D 75
  activating 31
  color 162
  copy 107, 120
  descriptor boxes 24
  label 33, 82, 141-142
  line style 73
  persistence 74
  recall 139, 141
  save 133, 141
  turn on/off 31
track 91, 94
TRC files 120
trend 91, 94-95
trigger 55
  automating 69
  controls 55, 57, 65
  counter 65, 161
  delay 48
  descriptor box 25
  holdoff 65
  software assisted 68
  time 50, 65
TriggerScan 69

U
UL compliance 170
undo 21
units 42, 114
utilities 153

V
vertical 40
  controls 41
  offset 41
  resolution 108
  sensitivity 40-41

W
waveform files 120, 134
  auto save 138
  recall 139
WaveScan 122
WaveStudio 163
WEEE 171
Windows
  dialogs 19
  networking 16
  power settings 15

X
XWEB 88
XY plots 73

Z
zoom 29, 31, 34
  controls 34, 163
  descriptor box 24
  multi-zoom 34
  undo 21



700 Chestnut Ridge Road
Chestnut Ridge, NY 10977
USA

teledynelecroy.com