TELEDYNE LECROY
Everywhereyoulook™

Operator's Manual

WaveRunner 6 Zi
HRO
Oscilloscopes

WaveRunner 6 Zi and HRO Oscilloscopes

Operator's Manual

September 2015

WaveRunner 6 Zi and HRO Oscilloscopes Operator's Manual

926367-00
September 2015

Contents

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.

The manual is arranged in the following manner:

- **Safety** contains important precautions and information relating to power and cooling.
- **Introduction** through **Maintenance** cover everything you need to know about the operation and care of the instrument.

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

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 Chief Technology Officer
Teledyne Lecroy

About This Manual

This manual documents the standard, shared functionality of WaveRunner 6 Zi and HRO oscilloscopes. Additional functionality delivered with various software option is documented in manuals available from teledynelecroy.com.

Similarly rated HRO models can be assumed wherever you see "WaveRunner" referred to in the text.

Safety Instructions

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 considerations:

CAUTION of potential damage to instrument or **WARNING** of potential bodily injury. Refer to the accompanying information. Do not proceed until the information is fully understood and conditions are met.

CAUTION, possibility of electric shock.

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

Comply with the following instructions 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 front 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.

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

Operating Environment

Temperature: 5 to 40° C.

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

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

Cooling

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

CAUTION. Do not block the cooling vents.

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

Cleaning

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

WARNING. 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.

Calibration

The oscilloscope is calibrated at the factory prior to being shipped. The recommended calibration interval is one year. Calibration should be performed by qualified personnel only.

Schedule an annual factory calibration as part of your regular maintenance. Extended warranty, calibration, and upgrade plans are available for purchase. Contact your Teledyne LeCroy sales representative or customersupport@teledynelecroy.com to purchase a service plan.

The oscilloscope software includes both automatic and user-initiated deskew calibration functions.

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

Power

AC Power Source

100-240 VAC (±10%) at 45-66 Hz
100-120 VAC (±10%) at 380-420 Hz
100-240 VAC (±10%) at 50/60 Hz

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

Measurement Category rating: CAT II*, 300V

* Refers to measurements performed on circuits directly connected to the low-voltage installation, per IEC/EN 61010-1.

Power Consumption

Nominal consumption: ≤ 400 W 400 VA)

Maximum consumption (active probes installed on all channels, peripherals connected to all USB ports, etc.): ≤ 500 W (500 VA)

Standby consumption: 15 W

Grounding

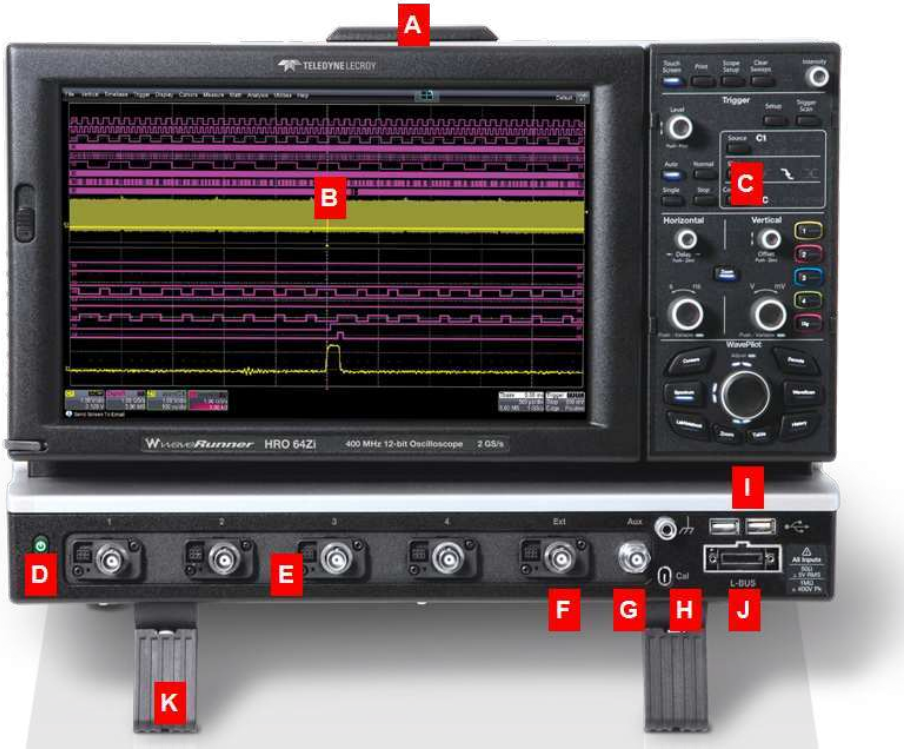
The WaveRunner is provided with a 10A/250V 18AWG rated grounded cord set containing a molded three-terminal polarized plug with an IEC-60320 (Type C13) connector. The connector mates to a compatible power inlet on the instrument for making line voltage and safety ground connections.

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.

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

Introduction to WaveRunner

Front of Oscilloscope



A. Built-in carrying handle
B. Rotating display panel
C. Front panel
D. Standby Power button
E. Analog input channels (C1-C4)
F. External trigger connector
G. Aux Out connector
H. Ground connector (top) and calibration hook (bottom)
I. USB ports (2)
J. LBUS connector
K. Tilting feet

Side of Oscilloscope

A. VGA connector (for external monitor)
B. Audio In/Out (Mic, Speaker, Aux In)
C. USB ports (2)
D. Ethernet port
E. USBTMC port
F. Optional accessory pouch

Back of Oscilloscope

A. AC power inlet
B. Optional accessory pouch (if installed)

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.

Use the **File > Shutdown** menu bar option to switch "off". Pressing Power again will execute a shutdown, but we do not recommend doing this because it does not allow the Windows operating system to shut down properly, and memories and setup panels will not be saved. Never power off by pulling the power cord from the socket or shutting off a connected power strip without first shutting down properly using File > Shutdown.

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

X-Stream Application Window

X-Stream (base application) runs on a Windows operating system and functions exactly as do other Windows applications.

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

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

Screen Saver

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

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

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 panel to a network access device.

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

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

USB Peripherals

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

Printer

The WaveRunner supports USB printers compatible with the instrument's Windows OS. Go to Utilities > Utilities Setup > Hardcopy to configure printer settings.

External Monitor

You may operate the instrument using the built-in touch screen or attach an external monitor for extended desktop operation. A properly configured external touch-screen monitor will take on all the touch-screen capabilities of the internal display.

NOTE: External monitors with Fujitsu touch-screen drivers can not be used to control the system, as this driver will conflict with the instrument's main display driver. These monitors may be used for display only.

Connect the monitor cable to on the of the instrument. Minimize the X-Stream application and use the Windows controls to configure the display. Configure the instrument as the primary monitor and be sure to extend, not duplicate, the display.

External Controller

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

Other Instrument (for Reference Clock)

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 > Reference Clock to configure the clock.

Other Auxiliary Device

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

Signal Interfaces

ProBus Interface

Analog input channels 1-4 are equipped with the ProBus interface.

The ProBus interface contains a 6-pin power and communication connection and a BNC signal connection to the probe. It offers both 50 Ω/1 MΩ input impedance and provides probe power and control for a wide range of probes such as high impedance passive probes, high impedance active probes, current probes, high voltage probes, and differential probes. ProBus also includes sense rings for detecting passive probes. The ProBus interface may also have a BNC-terminated cable connected directly to it.

ProBus is based on a BNC connector and, depending on the exact BNC connector used and the oscilloscope design, is rated for up to 4 GHz with 50 Ω coupling or up to 1 GHz for 1 MΩ coupling (depending on the exact model purchased).

Probes

Teledyne LeCroy offers a variety of probes for use with your oscilloscope. Visit teledynelecroy.com for a list of compatible probes and ordering information.

LBUS Interface

The LBUS (LeCroy Serial Bus) interface is designed to provide high-speed data transfer between Teledyne LeCroy oscilloscopes and other devices. It can be used with the optional:

- MS-250/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.

User Interface

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.

Miscellaneous Controls

This topmost section of the front panel provides the following functions:

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 Hardcopy 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.

Front Panel Trigger Controls

The Trigger front panel group corresponds to the Trigger dialog.

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. A fast trigger rate causes the light to stay lit continuously.

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 arms the oscilloscope to trigger once (single-shot acquisition) when the input signal meets the trigger conditions set for the type of trigger selected. If the scope is already armed, it will force a trigger.

Stop prevents the 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 Horizontal Controls

The Horizontal front panel group corresponds to the Timebase dialog.

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.

Front Panel Vertical Controls

The Vertical front panel group corresponds to the Channel dialog.

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.

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.

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.

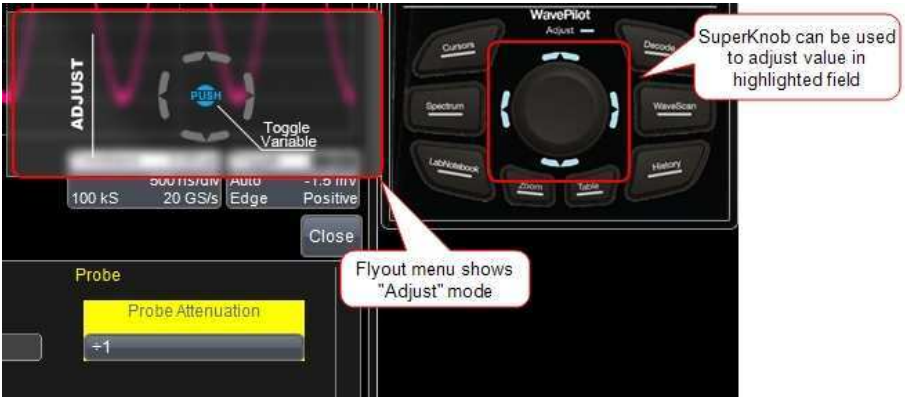
ADJUST

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.

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).

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.

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).

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.

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.

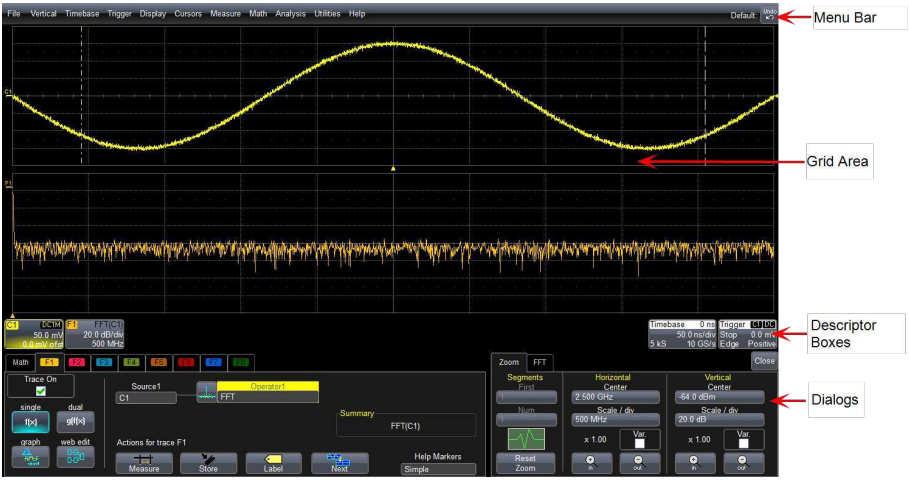
To tilt the angle of the display, firmly grab both sides of the panel. Pull the top of the panel toward you to tilt the display down; Push the top of the panel away from you to tilt the display up.

Touch Screen

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

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

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



Menu Bar

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

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

If an action can be “undone”, a small 

Undo button appears at the far right of the menu bar. Click this to return to the previous display.

Quick Access Toolbar

The Quick Access toolbar is located at the right side of the menu bar. You can use these buttons to quickly access trigger and acquisition processing functions, regardless of where you are in the XStream application.

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 arms the oscilloscope to trigger once (single-shot acquisition) when the input signal meets the trigger conditions set for the type of trigger selected. If the oscilloscope is already armed, it forces a trigger.

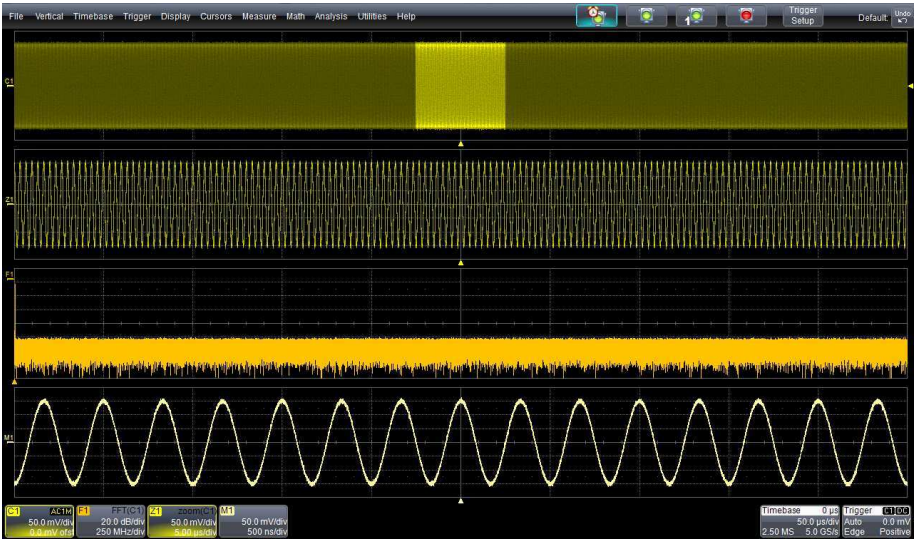
Stop prevents the oscilloscope from triggering on a signal. If you boot up the instrument with the trigger in **Stop** mode, the message "no trace available" is shown.

Trigger Setup opens the Trigger Setup dialog, same as the menu selection, **Trigger > Trigger Setup**.

Grid Area

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

The grid area can be divided into multiple grids showing different types and numbers of traces (by default, it will divide automatically as needed up to 16 times). Regardless of the number and orientation of grids, every grid always shows the same number of Vertical levels. Therefore, absolute Vertical measurement precision is maintained.



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

Adjusting Grid Brightness

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

Grid Indicators

These indicators appear around or on the grid to mark important points on the display. They are matched to the color of the trace to which they apply.

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

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

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

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

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

Grid Context Menu

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

Descriptor Boxes

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

- **Inform**—descriptors summarize the current trace settings and its activity status.
- **Navigate**—touch the descriptor box once to activate the trace; the box will be highlighted. Touch it a second time to open the trace setup dialog.
- **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, Gain Setting, Offset Setting, Sweeps Count (when Averaging), and Vertical Cursor positions.

Codes are used to indicate pre-processing that has been

applied to the input. The short form is used when several processes are in effect.

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

Timebase and Trigger Descriptor Box

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

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

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

Dialogs

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

Right-Hand Dialogs

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

Action Toolbar

Several setup dialogs contain a toolbar at the bottom of the dialog. These buttons 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.

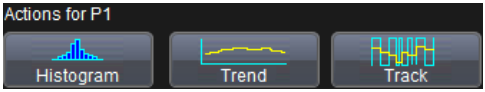
Store loads the active trace into the corresponding memory location (C1, F1 and Z1 to M1; C2, F2 and Z2 to M2, etc.).

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.



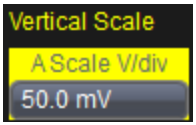
Message Bar

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 in this area.

Entering/Selecting Data

Touch & Type

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



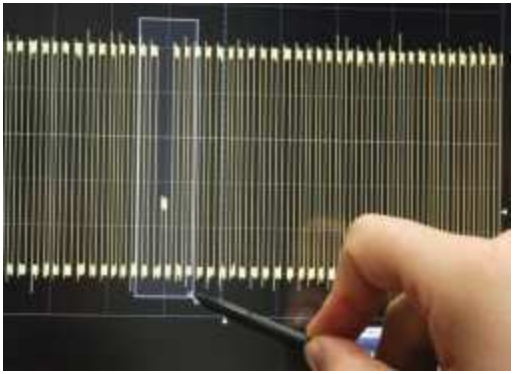
In other cases, data entry fields appear highlighted on the display. When a data entry field is highlighted, it is active and can be modified by using the **front panel Adjust knob**. If you have a keyboard installed, you can type your entry in the active field. Or, you can touch again, then make your entry on the pop-up.

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

Touch & Drag

Touch-and-drag waveforms, cursors, and trigger indicators to reposition them on the grid; this is the same as setting the values on the dialog. Use the setup dialogs to make exact entires.

Quickly zoom areas of the grid by touching and dragging to draw a selection box around a portion of the trace. Use the Zoom dialog controls to adjust the zoom exactly.



TOUCH & SWIPE

Touch and swipe the screen in an up or down direction to scroll long lists of values. You can also use scroll bars or Up/Down arrow keys to navigate to the desired value.

Printing/Screen Capture

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

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

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

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

Turning On/Off Traces

Channel Traces

From the menu bar, choose **Vertical > Channel <#> Setup** to turn on the trace. To turn it off, clear the **Trace On** checkbox on the C*x* dialog, or touch the trace and choose **Off** from the context menu.

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

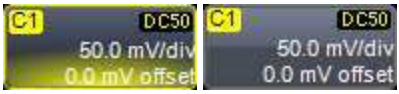
NOTE: The default is to display each trace in its own grid. Use the Display menu to change how traces are displayed.

Other Traces

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

Activate Trace

Although several traces may be open and appear on the grid, only one sampled trace (Cx) and one calculated trace (zoom, math function, or memory) is active and can be adjusted using front panel controls. Touch the trace descriptor box to activate the trace. A highlighted descriptor box indicates the trace is active. All actions now apply to that trace until you activate another.



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

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

Active descriptor box matches active dialog tab.

Moving Traces from Grid to Grid

You can move traces from grid to grid in several ways.

Next Grid Toolbar Button

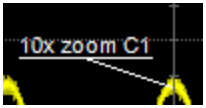
Open the Channel setup dialog for the trace you want to move, then touch the **Next Grid** Action toolbar button at the bottom of the dialog.

NOTE: If you have only one grid open, a second grid opens automatically when you select **Next Grid**.

Drag-and-Drop Descriptor Box

You can also 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.

Annotating 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. Touch the trace and choose **Set label...** from the context menu, or touch the **Label** Action toolbar button on the Cx 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 Traces

The Zoom function magnifies a selected region of a trace. Depending on your oscilloscope model, you can display up to eight or twelve zooms of any channel, math, or memory trace.

Zooms are created at the same vertical scale as the source trace and 10x horizontal magnification. You can adjust zooms the same as any other trace using the Front Panel Vertical and Horizontal knobs or the Zoom dialog touch screen controls.

The Multi-Zoom 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 Multi-Zoom.

Creating Zooms

QUICK ZOOM

Use the **front panel Zoom button** to quickly create one zoom trace for each displayed channel trace.

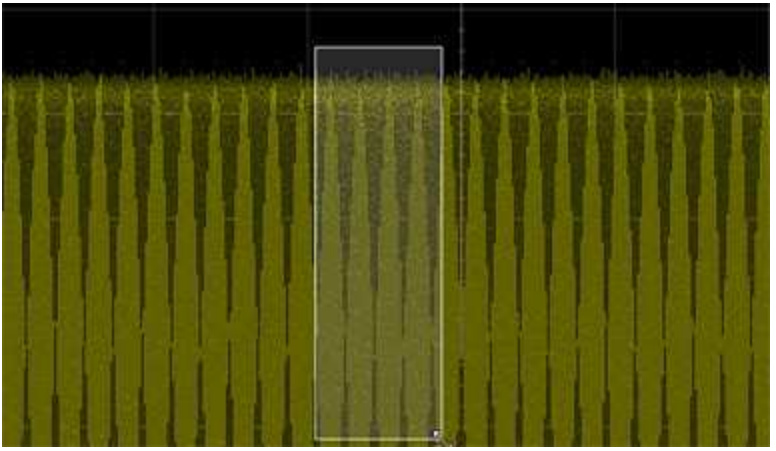
NOTE: Quick zooms are created at the same vertical scale as the source trace and 10x horizontal magnification.

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

MANUALLY CREATE ZOOM

To manually create a zoom, touch-and-drag to draw a selection box

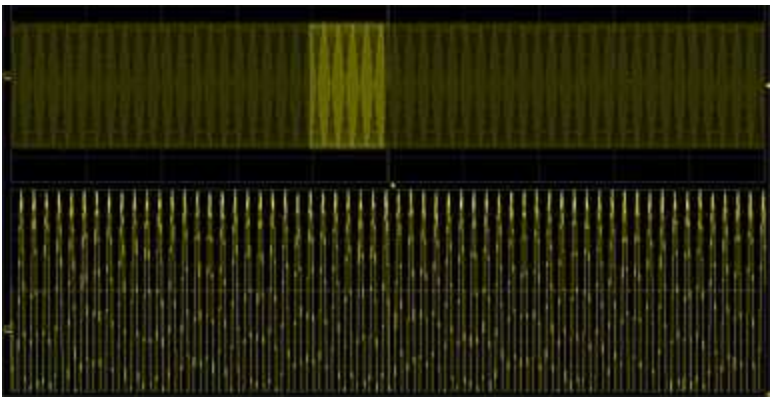
around any part of the source waveform.



Selection box over trace.

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

All zooms open in a new grid, or the next empty grid, with the zoomed portion of the source trace highlighted. If there are no more available grids, or you're using Single Grid, zooms will open in the same grid as the source trace.



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

SHOW/HIDE ZOOM

New zooms are turned on and visible by default. However, you can turn off a particular zoom if the display becomes too crowded, and the zoom

settings are saved in its Zx location, ready to be turned on again when desired.

To close the zoom, either:

- Touch the zoom descriptor box twice to open the Zoom dialog, then deselect **Trace On**.
- Touch the zoom trace to open the context menu, then choose **Off**.

ADJUST ZOOM

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

Zoom Controls

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

The main Zoom dialog contains selection boxes for turning on/off zoom traces. There are also options to:

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

Behind the main Zoom dialog is a separate tab for each potential zoom trace (Z1-Z*x*). Each dialog reflects the current scale settings 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

These controls on the **Z*x* dialogs** appear throughout the X-Stream software:

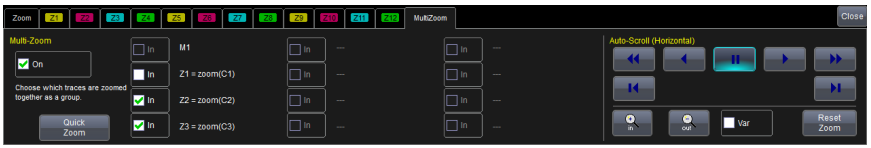
- **Out** and **In buttons** increase/decrease zoom magnification and consequently change the Horizontal and Vertical Scale settings. Touch either button until you've achieved the desired level.
- **Var. checkbox** enables zooming in single increments.
- **Horizontal Scale/div** sets the time represented 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.

Multi-Zoom

Multi-Zoom 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 MULTI-ZOOM

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



SCROLL WAVEFORMS

The Auto-Scroll controls appear at the right of the Multi-Zoom 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.

Vertical

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

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

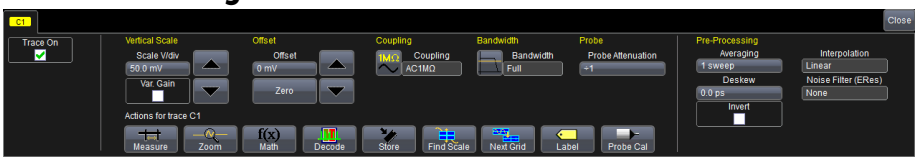
Vertical settings are made on the Channel dialog, labeled **C*x*** after the corresponding channel. To access it, choose **Vertical > Channel <#> Setup** from the menu bar, or touch the **Channel descriptor box**.

The C*x* dialog contains:

- Channel settings for scale, offset, coupling, bandwidth, and probe attenuation.
- Pre-processing settings for pre-acquisition processes that will affect the waveform, such as noise filtering and interpolation.

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

Channel Settings



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

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

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

A variety of **Bandwidth** filters are available at fixed settings. The exact settings vary by model. To limit bandwidth, select a filter from this field.

Coupling may be set to DC 50 Ω, DC1 M, AC1 M or GROUND (depending on model).

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.

Vertical Unit Override allows the units of the selected channel to be changed from Volts (V) to Amperes (A). This is useful when using a third-party current probe that is not auto-detected or when probing across a current sense resistor.

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 interpolation. 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 Vertical Deskew pre-processing setting and the Math deskew functions perform the same activity.

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. Use ERes on single-shot waveforms, or where the data record is slowly repetitive (when you cannot use averaging). Use it 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: zooming with high vertical gain, for example. For more information, see Enhanced Resolution.

Invert inverts the trace.

Probe Settings on Channel Dialog

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.

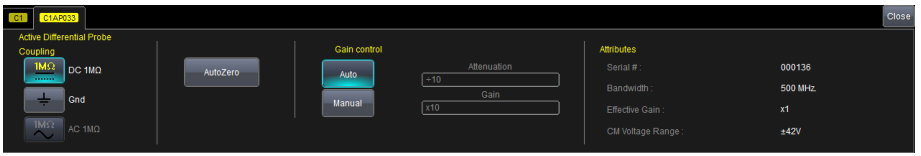
When a Teledyne LeCroy probe is connected to a channel input:

- Passive probe Attenuation is automatically set, and this field is disabled on the Channel setup dialog.
- A tab is added for active voltage and current probes to the right of the C*x* tab. Click on the tab to display the Probe dialog. The Attenuation field becomes a button to access the Probe dialog.

Channel dialog with tab for connected probe.

Probe Dialog Settings

The Probe Dialog immediately to the right of the C*x* 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.



Probe dialog showing the connected probe's control attributes.

Auto Zero Probe

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

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

DeGauss Probe

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

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

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.

Auto Setup

Auto Setup quickly configures the essential acquisition settings based on the first input signal it finds, starting with Channel 1. If nothing is connected to Channel 1, it searches Channel 2 and so forth until it finds a signal. Vertical Scale (V/div), Offset, Timebase (Time/div), and Trigger are set to an Edge trigger on the first, non-zero-level amplitude, with the entire waveform visible for at least 10 cycles over 10 horizontal divisions.

To run Auto Setup:

1. Either press the 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.

Cx Find Scale

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 **C*x* Find Scale** from the flyout menu.

Undo Auto Setup

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.

NOTE: You will undo all new "setup" work, such as new measurements or math function definitions entered since the Auto Setup, when you Undo the Auto Setup. Perform this work when the instrument is not in the Auto Setup mode if you wish for it to persist.

Restore Default Setup

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

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).

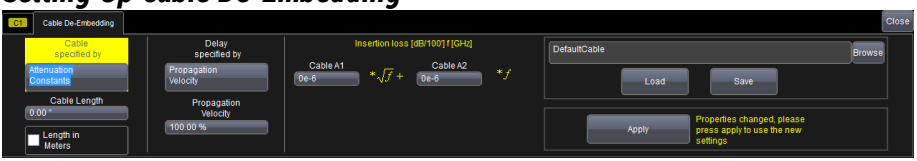
Cable De-Embedding Option

When making measurements on serial data signals, 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 your serial data measurements and potentially create mask test violations.

Cable De-Embedding, available as an option on WaveRunner 6 Zi and HRO oscilloscopes, 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.

When Cable De-Embedded is installed, each of the four channels has its own Cable De-Embedding dialog where you can individually describe the cable that is being used on each channel. Use it to enter either the attenuation table for the cable or two attenuation constants of the loss model for the cable (provided by the cable manufacturer). Then, specify the delay and physical length of the cable that you are using in your setup, and you are ready to view your corrected signal. When you remove the effects of the cable, the compensated signal is a more accurate representation of what was actually transmitted.

Setting Up Cable De-Embedding

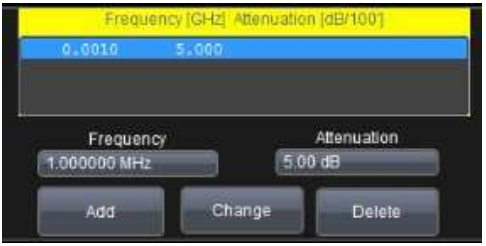


1. Touch **Vertical > Channel # Setup...** from the menu bar.
2. On the Cx dialog, check **Cable De-Embedding**to display the Cable De-Embedding tab.
3. On the Cable De-Embedding dialog, touch **Cable Specified by** and either:

   Choose Attenuation Constants, then enter the cable loss model constants (provided by the manufacturer) in **Cable A1** and **Cable A2**.

   Or

   Choose Attenuation Table, then enter the cable characteristics (provided by the manufacturer) in the table.



   Enter **Frequency** value and **Attenuation**values, then touch **Add** . 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 delay value in the field of the same name that appears below your selection.
5. Touch **Cable Length** and enter the length in inches.
6. Click **Apply** to initiate the cable de-embedding for the signal on this channel so that the effects of the cable are removed.

Saving Cable Configurations

Save cable configurations to easily load them at a later time. Cable configurations are not be preserved after exiting the application unless saved.

1. On the **Cable De-Embedding** dialog, touch **File Name [DefaultCable]** and enter a name, or touch **Browse** and select a file to overwrite.
2. Touch **Save**.

To load previously saved cable configurations, browse to the file, then touch **Load**.

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**.

The Timebase dialog contains settings for Sampling Mode, Timebase Mode, Real Time Memory, and Active Channels. There are related dialogs for Sequence Mode and Clock Source.

The Active Channels controls, located at the far right of the main Timebase dialog. By reducing the number of active channels from four to two, you can increase the sampling rate and memory of a single input by combining the digital capabilities of two channels.

Timebase Settings

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.

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

Set to Zero returns Delay to zero.

Real Time Memory

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

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

Set Maximum Memory 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 is a quick way to optimize 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

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

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 acquisition on Channels 1 and 4. 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.

NOTE: C1 can operate with either C3 or C4 at higher sample rate and memory since they belong to different pairs, and likewise C2. It just cannot operate with C2 without dropping the sample rate, nor can C3 operate with C4.

Refer to Acquisition Modes in the product datasheet for maximum sample rates.

Sampling Modes

Real Time Sampling Mode

Real Time sampling mode is a series of digitized voltage values sampled on the input signal at a uniform rate. These samples are displayed as a series of measured data values associated with a single trigger event. By default, the waveform is positioned so that the trigger event is time zero on the grid.

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

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

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

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

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

Sequence Sampling Mode

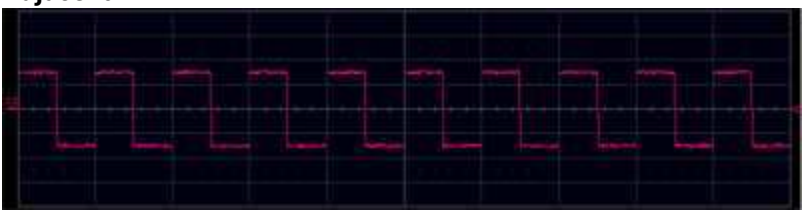
In Sequence Mode, the complete waveform consists of a number of fixed-size segments (see the instrument specifications at teledynelecroy.com for the limits). The instrument uses the sequence timebase setting to determine the capture duration of each segment as 10 x time/div. The desired number of segments,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

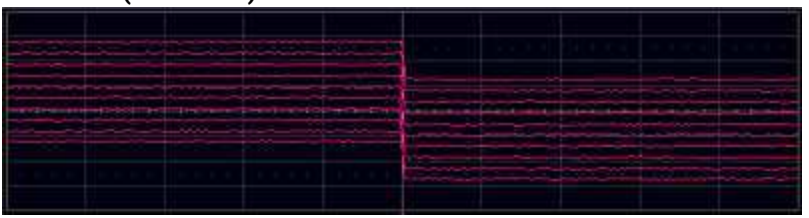
SEQUENCE DISPLAY MODES

There are five ways to display your segments:

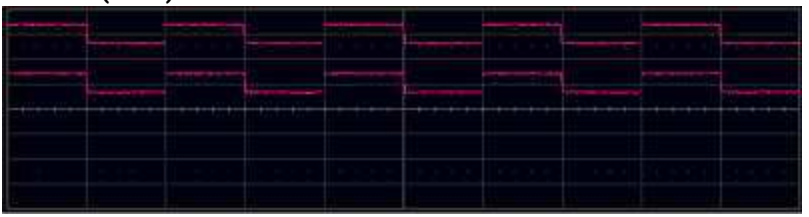
- **Adjacent**



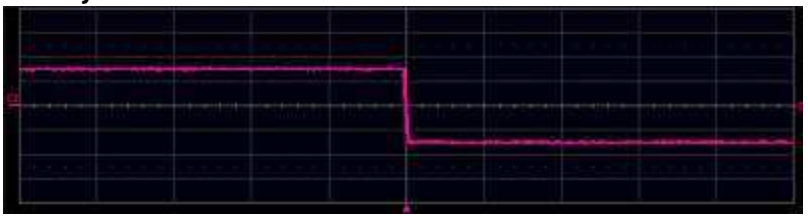
- **Waterfall (cascaded)**



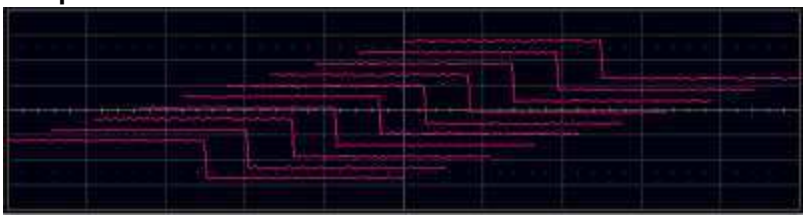
- **Mosaic (tiled)**



- **Overlay**



- **Perspective**



NOTE: Some display modes have limitations on the number of segments that can be shown at one time.

Set Up Sequence Mode

When setting up Sequence Mode, you define the number of fixed-size segments acquired in single-shot mode (see the instrument specifications for the limits). The instrument uses the sequence timebase setting to determine the capture duration of each segment. Along with this setting, the 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.

1. From the menu bar, choose **Timebase > Horizontal Setup...**.
2. Choose **Sequence Sampling Mode**.
3. On the **Sequence tab** under Acquisition Settings, touch **Number of Segments** and enter a value.

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

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

   NOTE: While optional, Timeout ensures that the acquisition completes in a reasonable amount of time and control is returned to the operator/controller without having to manually stop the acquisition.

5. Touch **Display mode** and select a sequence display mode from the

pop-up menu.

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

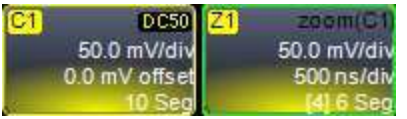
NOTE: Once acquisition has started, you can interrupt it at any time by pressing the **Stop** front panel button. In this case, the segments already acquired will be retained in memory.

View Segments in Sequence Mode

When in Sequence Mode, you can view individual segments easily using the Zoom trace dialog (Z*x*). A new zoom of the channel trace defaults to Segment 1. You can view later segments by changing the **First** segment to display and total **Num**(ber) of segments to display on the Z*x* 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. The Channel descriptor box reads 10. The Zoom descriptor box reads [4]3, meaning you are displaying a total of 3 segments, starting with segment 4.

Use the Zoom controls to change the scale factors of the trace.

View Segment as Math Function

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 (F*x*) 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** right-hand dialog 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.

VIEW SEGMENT TIME STAMPS

To view time stamps for each segment:

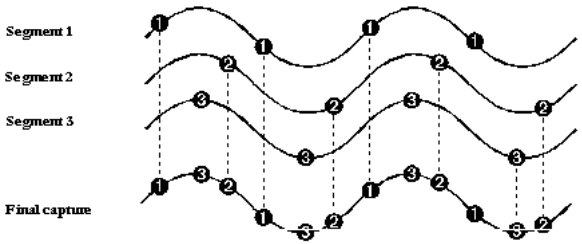
1. From the menu bar, choose **Timebase > Acquisition Status**.
2. Touch the **Trigger Time** tab.
3. Under **Show Status For**, choose **Time**.
4. In **Select Segment**, enter the segment number of interest.

   You can also touch the Up/Down Arrow buttons to scroll through segment times.

RIS Sampling Mode

RIS (Random Interleaved Sampling) allows effective sampling rates higher than the maximum single-shot sampling rate. It is used on repetitive waveforms with a stable trigger. The maximum effective RIS sampling rate is achieved by making multiple single-shot acquisitions at maximum real-time sample rate. The bins thus acquired are positioned approximately 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.

The instrument requires multiple triggers to complete an acquisition. The number depends on the sample rate: the higher the sample rate, the more triggers are required. It then interleaves these segments (as shown in the following illustration) to provide a waveform covering a time interval that is a multiple of the maximum single-shot sampling rate. However, the real-time interval over which the instrument collects the waveform data is much longer, and depends on the trigger rate and the amount of interleaving required.



Interleaving of sample in RIS sampling mode.

NOTE: RIS is not available when the oscilloscope is operating in fixed sample rate mode.

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

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

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

Clock Source Settings

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

Sample Clock

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

1. Connect a clock source of 50 Ohms, Ground, or 1 M Ohm impedance to the LBUS input using the optional WR6Zi-ExtRef-IN/OUT adapter.
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 input impedance.

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**.

Trigger

While the instrument continuously samples as long as a channel is turned on, it can only display up to its maximum memory in data samples. Triggers select an exact event/time in the waveform to display on the touch screen so that memory is not wasted on insignificant periods of the signal. The trigger may be set on a single channel, or a complex pattern of events across several channels of data.

On instruments with Mixed Signal capabilities, many trigger types can be set on either analog channels, including the External Trigger input, or digital lines. For digital triggering instructions, see the Operator's Manual for your Mixed Signal accessory.

For all trigger types, you can set:

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

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

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

Trigger Modes

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

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

In **Normal** mode, a sweep occurs only when the input signal achieves the trigger conditions. Otherwise it continues to display the last acquired waveform.

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

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

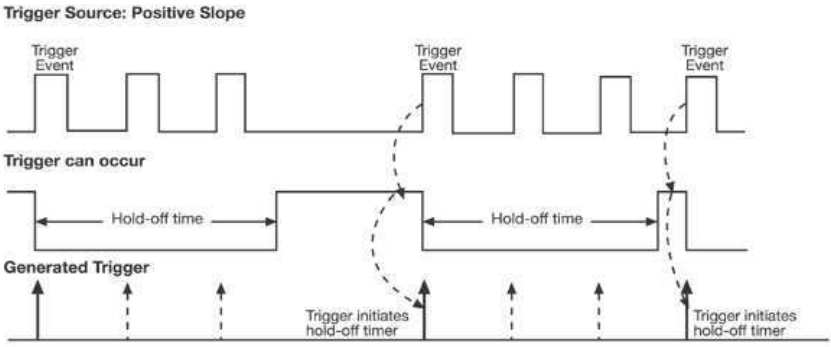
Trigger Holdoff

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, until the holdoff conditions are also met. The trigger fires when the holdoff has elapsed.

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

Hold Off by Time

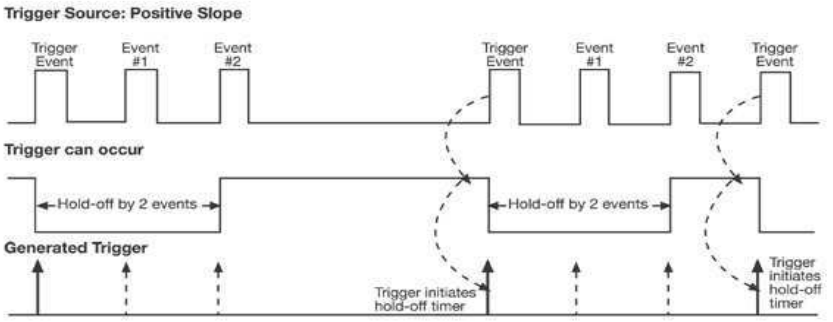
This is a period of time to wait to fire the trigger, either since the beginning of the acquisition or since the trigger conditions were met. You may achieve a stable display of complex, repetitive waveforms by placing a holdoff condition on the time between successive Edge trigger events. This time would otherwise be limited only by the input signal, the coupling, and the instrument's bandwidth. Select a positive or negative slope, and a minimum time between triggers.



Positive Edge trigger with holdoff by time. The broken up-arrows indicate potential triggers which would occur without holdoff. The bold arrows indicate where triggers actually occur when the holdoff has been satisfied.

Hold Off by Events

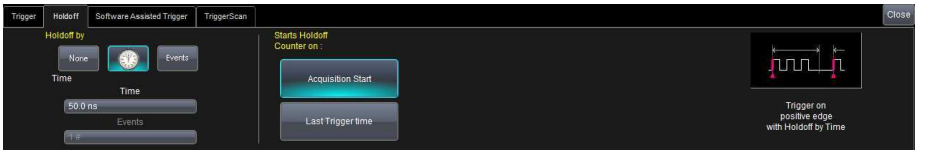
For purposes of Hold Off, Events refers to the number of times the trigger conditions have been met, counted either from the beginning of the acquisition or since the last trigger. For example, if the hold-off is two events counted from the beginning of the acquisition, the trigger fires on the third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



Positive Edge trigger with holdoff by events. The broken up-arrows indicate potential triggers which would occur without holdoff. The bold arrows indicate where triggers actually occur when the holdoff has been satisfied.

Holdoff Settings

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



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

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

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

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

Setting Up Triggers

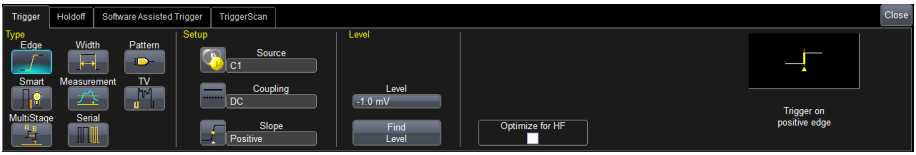
To access the Trigger setup dialogs, press the **front panel Trigger Setup** button or touch the **Trigger descriptor box**.

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

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

Edge Trigger

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



1. On the Trigger dialog, select **Edge** trigger type.
2. Choose the **Source** signal input.
3. Choose the type of signal **Coupling** at the input. Choices are:
   - **DC** - All the signal's frequency components are coupled to the trigger circuit for high frequency bursts or where the use of AC coupling would shift the effective trigger level.
   - **AC** - The signal is capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
   - **LFREJ** - The signal is coupled through a capacitive high-pass filter network, DC is rejected and signal frequencies below 50 kHz are attenuated. For stable triggering on medium to high frequency signals.
   - **HFREJ** - Signals are DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz (used for triggering on low frequencies).
4. Choose the **Slope** (edge) of the wave on which to trigger.
5. Enter the voltage **Level** upon which to trigger. The **Find Level** button sets the Level to the signal mean.

Width Trigger

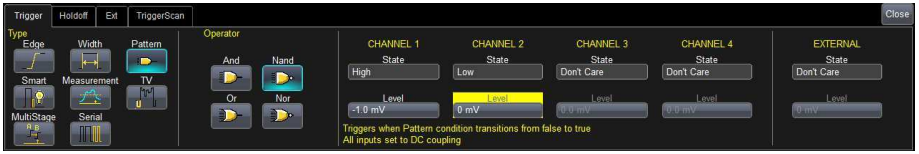
Width triggers upon finding a positive- or negative-going pulse width when measured at the specified voltage level.

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

Pattern Trigger

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

Pattern is the default trigger when the Mixed Signal option is connected to the instrument, as this configuration is common to users wish to find and trigger upon digital logic patterns. See the *MS-250 Operator's Manual* or *MS-500 Operator's Manual* delivered with your Mixed Signal option for instructions on setting up a digital pattern trigger.



1. On the Trigger dialog, select **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 NAND 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 **Attenuation**.

Window Trigger

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

1. On the Trigger dialog, select **Smart** trigger type, then choose **Window**.
2. Choose the **Source** signal input.
3. Choose the type of signal **Coupling** at the input. Choices are:
   - **DC** - All the signal's frequency components are coupled to the trigger circuit for high frequency bursts or where the use of AC coupling would shift the effective trigger level.
   - **AC** - The signal is capacitively coupled. DC levels are rejected, and frequencies below 50 Hz are attenuated.
   - **LFREJ** - The signal is coupled through a capacitive high-pass filter network, DC is rejected and signal frequencies below 50 kHz are attenuated. For stable triggering on medium to high frequency signals.
   - **HFREJ** - Signals are DC coupled to the trigger circuit, and a low-pass filter network attenuates frequencies above 50 kHz (used for triggering on low frequencies).
4. Choose to define the window using **Absolute** or **Relative** voltage levels.
5. If Absolute, enter the voltage **Upper Level** and **Lower Level**.

   If Relative, enter a **Nominal Level** plus or minus a **Delta** voltage.

Interval Trigger

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

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

Glitch Trigger

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

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

Dropout Trigger

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

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

Runt Trigger

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

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

SlewRate Trigger

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

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

Measurement Trigger

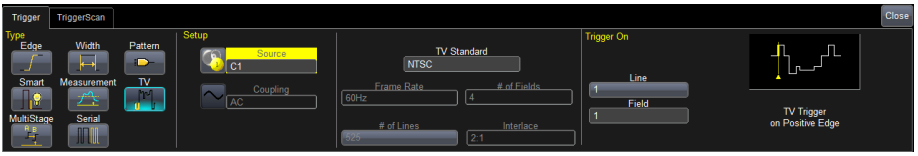
Measurement fires when the trigger source achieves the specified measurement. The available measurements depend on your model instrument and the options installed, and each will offer a different method for specifying the trigger condition, based on the type of measurement. Generally, you will be able to specify a triggering value or range of values, and for @level parameters, the voltage level at which the measurement must occur.

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

TV Trigger

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

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

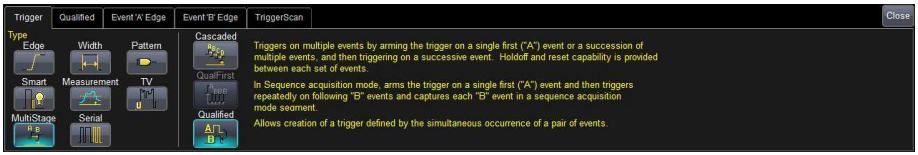


1. Choose the **Source** signal input.
2. Choose the signal **TV Standard**. To use a custom signal, also enter the:
   - **Frame Rate**
   - **# of Fields** per line
   - **# of Lines**
   - **Interlace** ratio
3. Choose the **Line** and **Field** upon which to trigger.

Qualified Trigger

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

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



Then, on the **Qualified dialog** choose the A and B events.

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

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

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

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

Serial Triggers

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

A 1.125 or 3.125 Gb/s 8b/10b Serial Trigger hardware option is available for WaveRunner oscilloscopes. (Speed depends on model; not available on

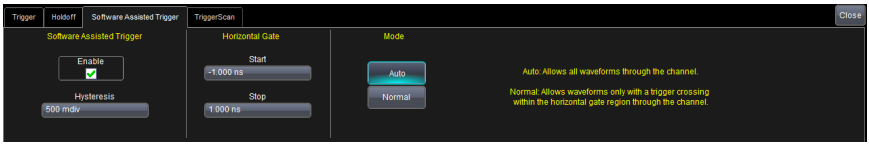
HRO.) This option provides capability for 8b/10b symbol triggering. The trigger hardware is factory installed and is permanently connected internally to C4. Complete information on the operation of this trigger can be obtained in the *High Speed Serial Triggers Instruction Manual* on the Teledyne LeCroy website.

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. Software Assisted Trigger provides a quick way to create eye diagrams.

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

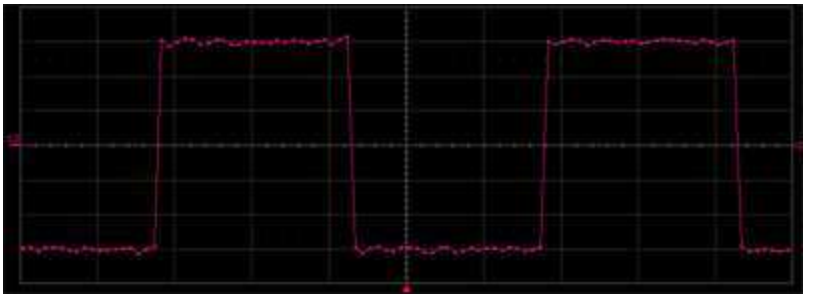
1. From the menu, choose **Triggers > Trigger Setup**, then touch the **Software Assisted Trigger tab**.



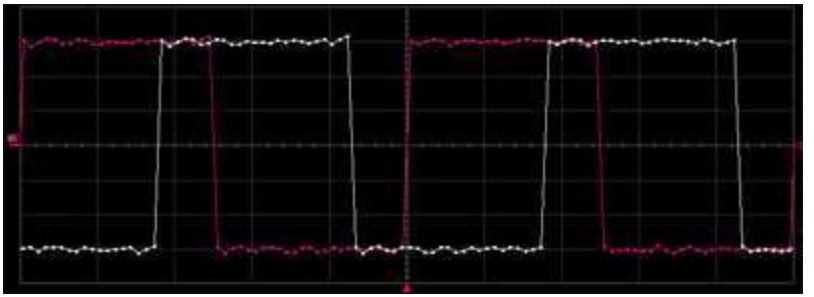
2. Touch **Enable**.
3. Create a trigger window by entering a **Hysteresis** value. This value sets a boundary above and below the main trigger level to exclude noise.
4. Choose **Auto** or **Normal**; this determines the trigger behavior when trigger crossings are not found in the trigger source waveform.
   - **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.

Example

The following waveform figure has software-assisted triggering disabled:

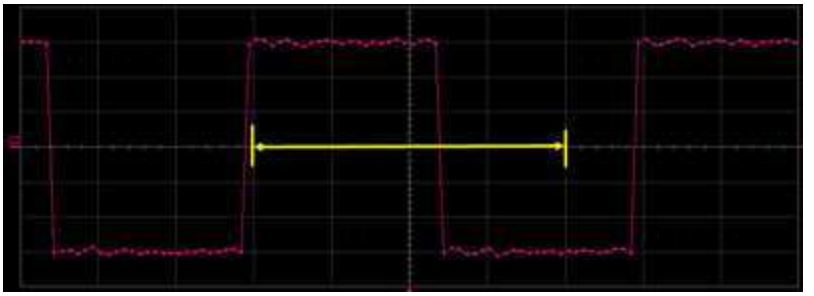


The next waveform figure has software-assisted triggering enabled. The red trace is aligned with the trigger point, indicated by the red marker below the grid. The white trace is the waveform at the input to software-assisted trigger processing. Notice how the white trace is the same as the previous figure.



The next figure shows a waveform outside the horizontal gate region (the closest positive edge is not within ±1 ns). The **Auto** and **Normal** difference is as follows:

- In **Auto** software assisted trigger mode, the waveform would be displayed, but not aligned.
- In **Normal** mode, it would not be displayed.



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 all functions to find the root cause of these anomalies including 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.

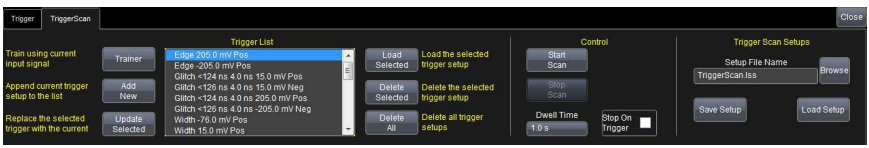
NOTE: Run the Trainer if you want to change the trigger types or if you change the channel or signal. You must acquire and display at least 3 cycles of a signal before running the Trainer.

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.

Modify 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 the **Stop On Trigger** checkbox. You can use this to isolate events as the 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.

Display

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

Auto Grid 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 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.

Display Settings

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

Grid Mode

Touch the **Grid button**, then select one of the grid modes.

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

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.

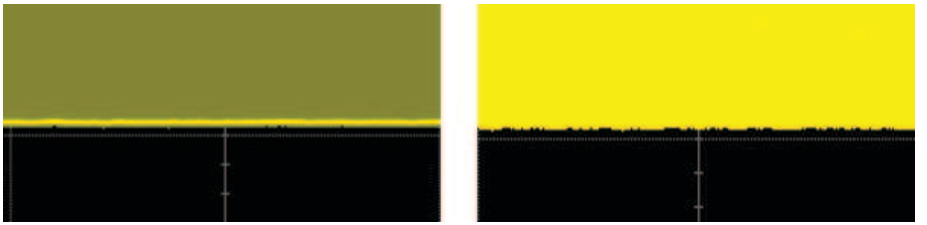
NOTE: Some waveforms may be hidden from view with the grid on top.

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

Trace

Choose a line style for 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

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

If you are using Sequence sampling mode, choose the display style.

Persistence

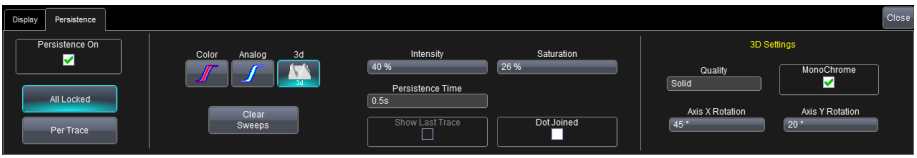
The Persistence feature retains waveform traces on the display for a set amount of time before allowing them to gradually "decay," similar to the display of old phosphor screen oscilloscopes. Use Persistence to accumulate on-screen points from many acquisitions to see your signal change over time. The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

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

To set up all traces together, touch **All Locked**. This constrains all input channels to the same persistence settings.

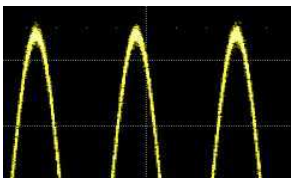
To set up traces individually, touch **Per Trace**.

To turn off persistence, clear **Persistence On** or select an individual trace's **None** (left-most) persistence mode button.



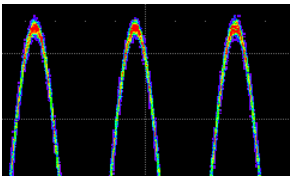
Persistence Mode

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



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

The Analog persistence view highlights the distribution of data so that you can examine it in detail.



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

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:

In **solid** quality, saturation is set at 50%, with brighter areas (or hotter colors) indicating highest intensity. The display can be either color or monochrome.

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, you can also turn the **X Axis Rotation** and **Y Axis Roation** of the waveform through 180° of rotation from -90° to +90°. In the examples above, the X-axis is rotated 60° an the Y-axis 15°.

TIP: Grab a corner of the persistence map and drag it in the desired direction.

Other Persistence Settings

Besides the different modes, you can select a **Saturation** level as a percentage of the maximum population. All populations above the saturation population are then assigned the highest color intensity: that is, they are saturated. At the same time, all populations below the saturation level are assigned the remaining intensities. Data populations are dynamically updated as data from new acquisitions is accumulated. A saturation level of 100% spreads the intensity variation across the entire distribution; at lower saturation levels the intensity will saturate (become brighter) at the percentage value specified. Lowering this percentage causes the pixels to be saturated at a lower population and makes visible those events rarely seen at higher saturation levels.

Intensity duplicates the Trace Intensity function on the main Display dialog.

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

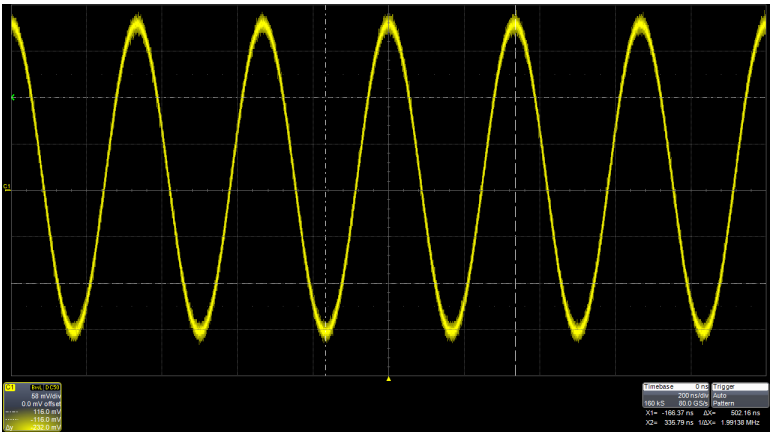
Choose to superimpose the last waveform over the persistence display by selecting **Show Last Trace** (not available in 3d mode).

To display traces as a continuous line (instead of a series of sample points), select **Dot Joined**.

Cursors

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

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



Horizontal and vertical relative cursors.

Cursor Types

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

Horizontal (Time) cursors intersect a point 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 depends on the Show option selected.

Vertical (Amplitude) cursors intersect a point 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. The readout depends on the Show option selected.

The **Both Rel**(ative) option places both relative cursors together.

Cursors on Math Functions

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 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.

Cursor Settings

Turn On/Off Cursors

To quickly turn on/off cursors, either:

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

Position Cursors

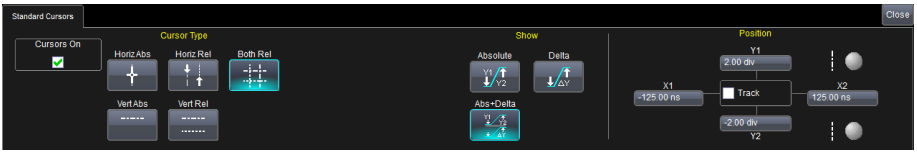
Te easiest way to position a cursor is to touch-and-drag the cursor line to a new position.

Alternatively, with the cursor on, turn the front panel **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.

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

Standard Cursors Dialog

These controls can be used instead of the front panel controls to set cursors or to refine the cursor 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 appear on the trace descriptor box 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 specific voltages and the difference between the specific voltages at the cursor locations.
- **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.

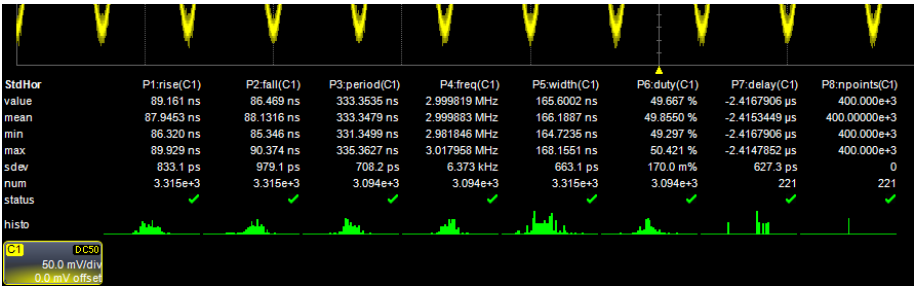
- **X 1** (negative) and **X 2** (positive) time from the zero point.
- **Y 1** (negative) and **Y 2** (positive) number of divisions from the zero level. May be a fraction of a division.
- **Track** locks cursor lines so they move together, maintaining their same relative distance from each other.

Measure

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

The instrument offers a quick selection of standard horizontal and vertical measurements. You can also create a custom set of parameters drawn from all available measurements.

Measurement readouts appear in a table below the grid. The **value** row shows the measurement taken on the last cycle of the last acquisition.



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

Symbol	Description
	Problem with the signal or the setup. Touch cell and see explanation in the 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.

Quick Measurements

Standard parameter sets are available for quick display. From the menu bar, choose:

- **Measure > Standard Horizontal** for a full set of common time parameters: freq, period, width, rise, fall, delay, duty, num points.
- **Measure > Standard Vertical** for a full set of common voltage parameters: mean, sdev, max., min., ampl, pkpk, top, base.

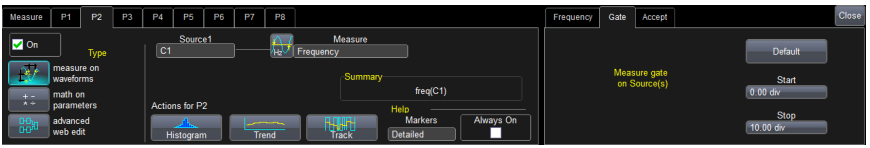
Setting Up Custom Measurements

To configure a custom set of measurement parameters:

1. From the menu bar, choose **Measure > Measure Setup**. 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 **P*x* tab** of an unused location (or one that you want to change).



4. Select the Type **Measure On Waveforms**.
5. Touch **Source1** and select the channel, math trace, memory trace, or other waveform to be measured.
6. Touch the **Measure** field and select the measurement from the pop-up menu.
7. Make any further selections on the right-hand dialogs that appear after your Measure selection. These are explained on the dialog and are sometimes necessary to fully define the selected measurement.
8. Optionally, turn on Statistics, Histicons, or Help Markers, or use the right-hand dialogs to gate or qualify measurements.
9. Check **On** to enable the parameter and add it to the measurement readout table.

Viewing Statistics

You can add the statistical measures mean, min., max., sdev, and num(ber of measurements computed) to the table by checking **Statistics On**, or by choosing **Measure > Statistics** from the menu bar.

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.

Viewing 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.

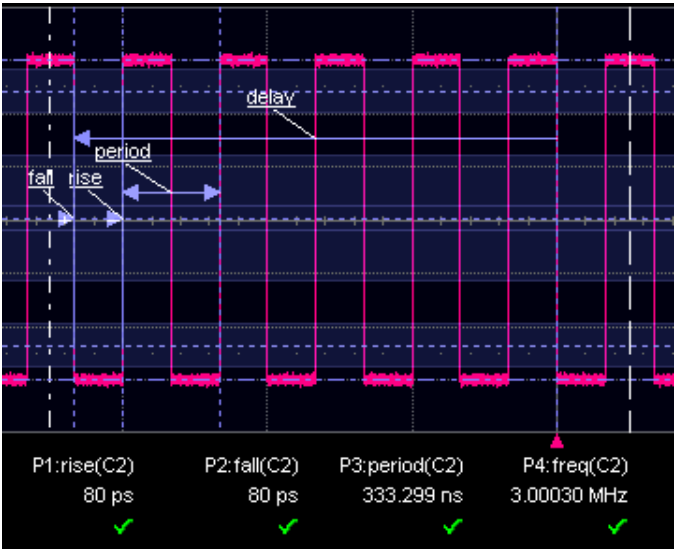
Help Markers

Help Markers clarify measurements by displaying cursor lines and labels marking the points being measured on the trace. For at-level parameters, markers make it easier to see where your waveform intersects the chosen level. This feature also displays any hysteresis band that you have set about that level.

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

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.

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.



Detailed Help Markers on standard horizontal parameters.

Level and Slope

For several time-based measurements, you can choose to begin the measurement on positive, negative, or both slopes. For two-input parameters, such as Dtime@level, you can specify the slope for each input, as well as the level and type (percent or absolute).

Make Level selection on the right-hand **Level dialog** when it appears.

Gating Measurements

By using gates, you can narrow the span of the waveform on which to perform parameter measurements, allowing you to focus on the area of greatest interest. For example, if you "gate" five rising edges of the waveform, 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 by specifying a position down to hundredths of a division using the **Gate Start** and **Stop** fields on the Gate right-hand dialog.

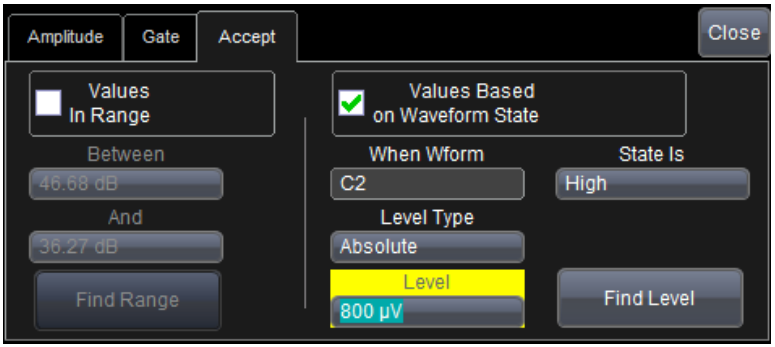
Touch the **Default** button to return gates to the width of the trace.

Qualified Measurements

Some measurements can be constrained to a vertically or horizontally limited range, or to occurrences 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 right-hand dialog 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** right-hand dialog, 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: Depending on whether it is a vertical or horizontal measurement, the correct units are automatically displayed (V, s, Hz, dB) in Between and And. If you select a simple ratio parameter (such as power factor) that yields a dimensionless number, no units will be displayed.

Waveform Gated 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** right-hand dialog, 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.

Math on Parameters

In addition to waveform measurements, you can set up a custom parameter that performs mathematical operations on measurements.

The setup for Math on Parameters is much like that for other custom parameters. Some parameters can also be qualified on the Accept dialog using value ranges or gating waveforms, as can regular waveform measurements.

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.

Exclusions

The parameter math feature prevents multiplication and division of parameters that return logarithmic values.

Parameters that are already the result of parameter math operations are also excluded. If they are included in a remote control setup command, an error message is generated and the setup canceled.

Set 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. These should be other parameters than that you are now using for Math on Parameters.
6. Check **On** to enable the new output parameter and add it to the measurement readout.

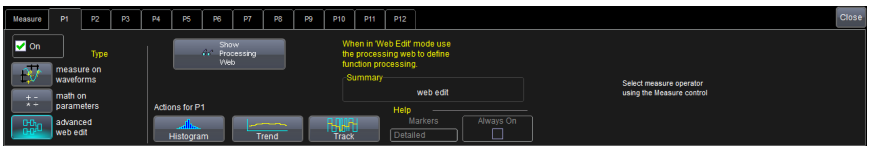
Using Web Edit

Whereas with Math on Parameters you can apply a single math operation to a measurement parameter, the Web Edit software (available as an option on WaveRunner) enables you to create custom processes that potentially chain many operations and apply them to the output of a single parameter or math function. 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 parameter value shown in the Measure table represents the numeric result of this process flow applied to the raw, acquired sample. Math functions would likewise display the waveform trace that results from the process.

Create Processing Web

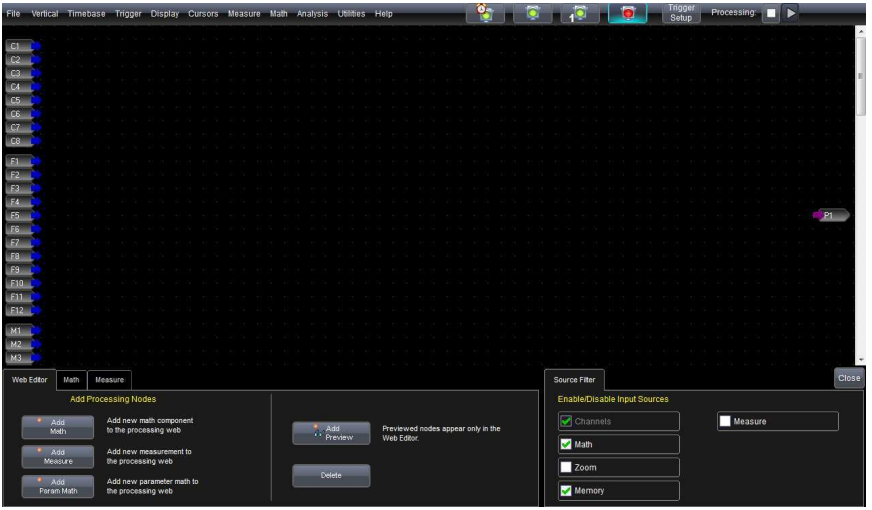
1. From the menu bar, choose **Measure > Measure Setup**.
2. On the Measure dialog, choose Measure Mode **My Measure**.
3. Open the **P*x*** dialog of an unused parameter (or one that you want to change) and select **Advanced Web Edit**.



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.

TIP: On the Source Filter right-hand dialog, 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.



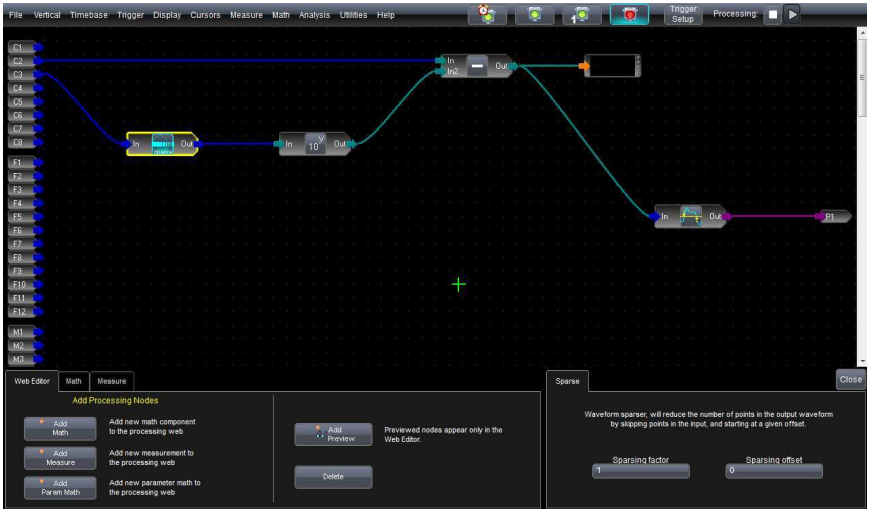
5. On the Wed Editor dialog, choose the type of process to add to the flow. You can **Add Math**, **Add Measure**, or **Add Param Math** (Math on Parameters).
6. From the pop-up, choose the operation or measurement that will occur at that node. A new block appears on the display. Drag the block to a 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 they are moved.

7. 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.



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

10. Drag the output pin from the final process block to the input pin on the terminal to complete the flow.

Preview Process

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.

List of Standard Measurements

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

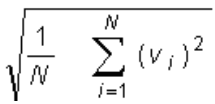
NOTE: There may be additional measurements available depending on the software options installed.

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

<table>
<tr><th>Measurement</th><th>Description</th></tr>
<tr><td>Duration
(dur)</td><td>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.</td></tr>
<tr><td>Duty Cycle</td><td>Percent of period for which data are above or below the 50% level of the signal.</td></tr>
<tr><td>Duty@level
(duty@lv)</td><td>Percent of period for which data are above or below a specified level.</td></tr>
<tr><td>Dwidth@level</td><td>Difference (delta) between adjacent widths above or below a specified level.</td></tr>
<tr><td>Edge@level
(edge@lv)</td><td>Number of positive edges in waveform that cross the specified threshold level.</td></tr>
<tr><td>Edge to Edge</td><td>Delta time between some edge on the first source to some (other) edge on a second source.</td></tr>
<tr><td>Fall 80-20%
(fall8020)</td><td>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.</td></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</td></tr>
</table>

Measurement	Description
	period measurement is calculated as the frequency.
Freq@level (freq@lv)	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.
FWHM	Measures the width of the largest area histogram peak at half of the populationof the highest peak.
FWxx	Measures the width of the largest area histogram peak at xx% of the population of the highest peak.
Half Period (hper)	Half period of a waveform.
Hist ampl (hampl)	Difference in value between the two most populated peaks in a histogram.
Hist base	Value of the left-most of the two most populated histogram peaks.
Hist max pop	Peak with maximum population in a histogram.
Hist maximum	Value of the highest (right-most) populated bin in a histogram.
Hist mean	Average or mean value of data in a histogram.
Hist median	Value of the 'x' axis of a histogram that divides the population into two equal halves.
Hist minimum	Value of the lowest (left-most) populated bin in a histogram.
Hist mode	Position of the highest histogram peak.
Hist pop@x	Population at bin for specified horizontal coordinate.
Hist range	Calculates range (max-min) of a histogram.
Hist rms	Root mean square of the values in a histogram.
Hist sdev	Standard deviation of values in a histogram.
Hist top	Value of the right-most of the two most populated histogram peaks.
Hist X@peak	Value of the *n*th highest histogram peak. You supply the value of *n*.
Hold Time	Time from the clock edge to the data edge.
Last	Time from trigger to last (rightmost) cursor.
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**

Measurement	Description
	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.

Measurement	Description
Period	The time between every other pair of 50% crossings. Starting with first transition after left measurement gate, period is measured for each transition pair, with values averaged to give final result.
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.
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	Standard deviation of the data between the measure gates using the

Measurement	Description
(sdev)	formula: $\sqrt{\frac{1}{N}\sum_{i=1}^{N}(v_i - mean)^2}$ Where: vi denotes measured sample values, and N = number of data points within the periods found up to maximum of 100 periods. This is equivalent to the rms for a zero-mean waveform. Also referred to as AC RMS
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.
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.

Graphing Measurements

Measurements can be viewed in several 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 are graphical representations of data which divide it into intervals, or bins. These intervals/bins are plotted on a bar chart such that the bar height relates to the number of data points within each bin.

Histograms can be created to visualize the results of measurement parameters or math functions. Regardless of the source, the histogram is created as a Histogram function (Fx) trace, and the number of sweeps (k#) comprising the histogram is shown on the function descriptor box. The Phistogram function creates a histogram of a persistence display.

The range of a histogram is limited to the portion of the (measurement or math) source trace that is visible on screen. If you zoom in on a trace, the histogram does not contain data for the no longer visible parts of the original trace.

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

CREATE HISTOGRAM

1. Open the parameter setup (P*x*) dialog for the measurement you wish to histogram.

   TIP: A quick way to do this is to touch the measurement table cell.

2. Touch the **Histogram** toolbar button at the bottom of the P*x* dialog and choose the function location (**F*x***) in which to display the histogram. The histogram opens in a new grid along with its function descriptor box.
3. Touch the new **hist descriptor box** to display the F*x* dialog, then open the **Histogram right-hand dialog**.

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.

NOTE: The histogram can also be created using the Math Setup dialog. First choose the function location (Fx), then choose the Source parameter and the Histogram Operator.

Create Persistence Histogram

You can create a histogram of a persistence display, which graphs a horizontal or vertical “slice” of a waveform.

NOTE: This operation is different than the Histogram math operation and is not affected by Center and Width settings made on any existing Histograms.

1. Choose **Math > Math Setup...** from the menu bar to access the Math dialog.
2. Touch an open **Fx button** and select **Phistogram** from the pop-up menu.
3. Touch the **Fx tab** to open the Function setup dialog, then select the **Source1** trace from the pop-up.
4. Open the **Phistogram** right-hand dialog.
5. Touch **Slice Center** and use the pop-up keypad to enter a value.
6. Touch **Slice Width** and use the pop-up keypad to enter a value.
7. Touch **Slice Direction** and select Horizontal or Vertical slice from the pop-up menu.

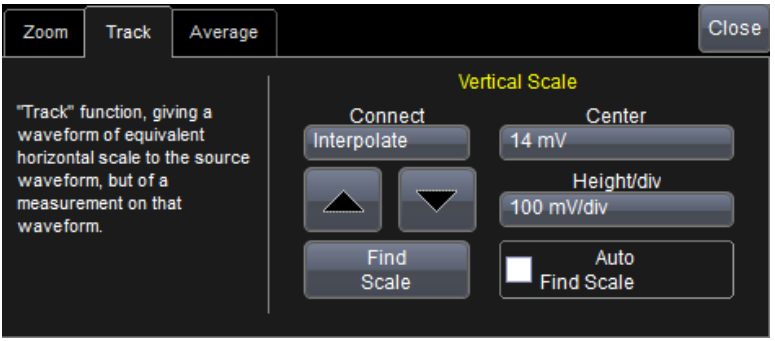
Track

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. 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.

   TIP: A quick way to do this is to touch the measurement table cell.

2. Touch the **Track** toolbar button at the bottom of the Px dialog and choose the function location (**F1-F12**) in which to display 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 right-hand dialog**.



4. On the Track right-hand dialog, **uncheck Auto Find Scale** and enter a new **Center** and **Height/div**. You can also use **Find Scale** to automatically find suitable values.
5. Choose a line **Connect** function of either Interpolate or Extend.

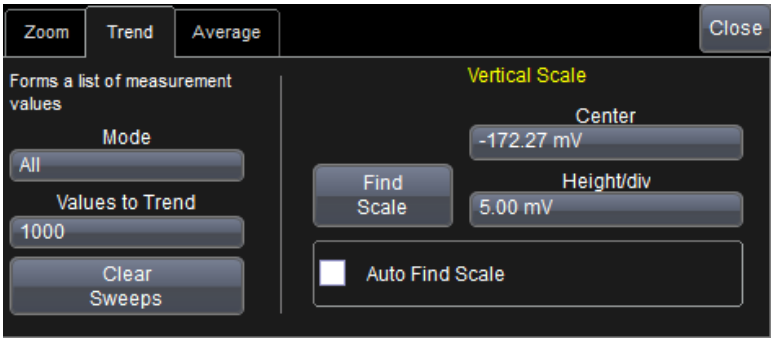
Trend

A Trend is a plot composed of a series of parameter measurements in the order the measurements were taken. The vertical units are those of the source parameter, the horizontal unit is measurement number. The Trend contains a single value for each measurement. Trends are especially useful for visualizing the history of a parameter over an extended period of time or over multiple acquisitions.

1. Open the parameter setup (P*x*) dialog for the measurement you wish to histogram.

   TIP: A quick way to do this is to touch the measurement table cell.

2. Touch the **Trend** toolbar button at the bottom of the Px dialog and choose the function location (**F1-F12**) in which to display 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 right-hand dialog**.



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, uncheck **Auto Find Scale** and enter the new **Center** and **Height/div** values. You can also use **Find Scale** to automatically find suitable values.

Track vs. Trend

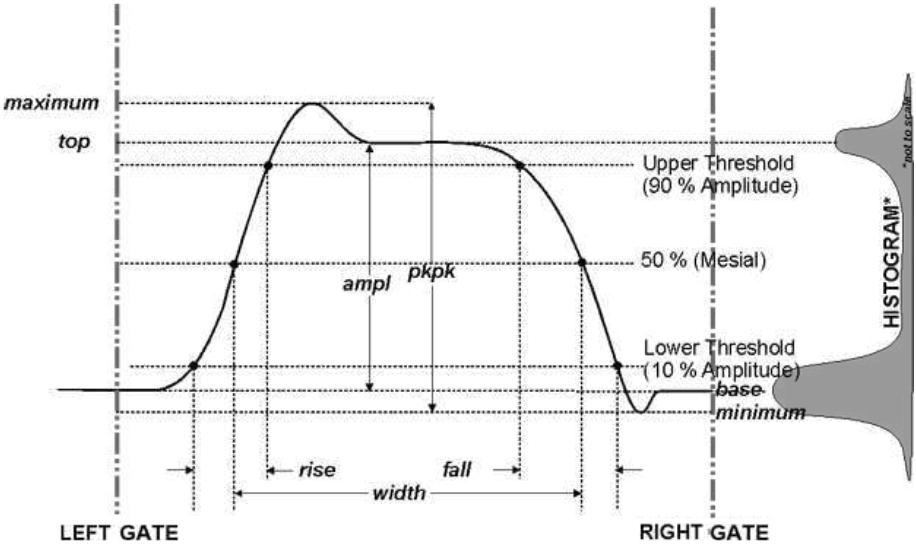
Both Track and Trend are tools that can be used to plot measurement data and observe variations with respect to time. The table below summarizes their differences. In general, Track is the tool to use if you want to capture a continuous stream of data spaced closely together. To understand the change in a parameter with time, Trend can be used if your data is spaced widely apart and longer than the dead-time between acquisitions. Think of Trend as a strip chart recorder for your instrument.

Characteristic	Track	Trend
Representation	Parameter value vs. time	Parameter value vs. event
Behavior	Non-cumulative (resets after every acquisition). Unlimited number of events	Cumulative over several acquisitions up to 1 million events
Time Correlation to Other Data	Yes	No
Monitors an Evolution in the Frequency Domain	Yes	No. Trend points are not evenly spaced in time and therefore cannot be used for an FFT.
Monitors the Evolution of a Measurement Parameter over Several Acquisitions	No. Track resets after every acquisition.	Yes
Ensures No Lost Measurement Data	Yes. Maximum time period that can be captured is limited by acquisition memory and sampling rate.	No. Since data can be accumulated over many acquisitions, and since the instrument takes time to calculate measurement values and to display data before the trigger is re-armed, data can be missed.

How the Oscilloscope Calculates Measurements

Determining Top and Base Lines

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



Determining Rise and Fall Times

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

Threshold levels for rise or fall time can also be selected using absolute or relative settings (if @level measurements are included with your model). If 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. But 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

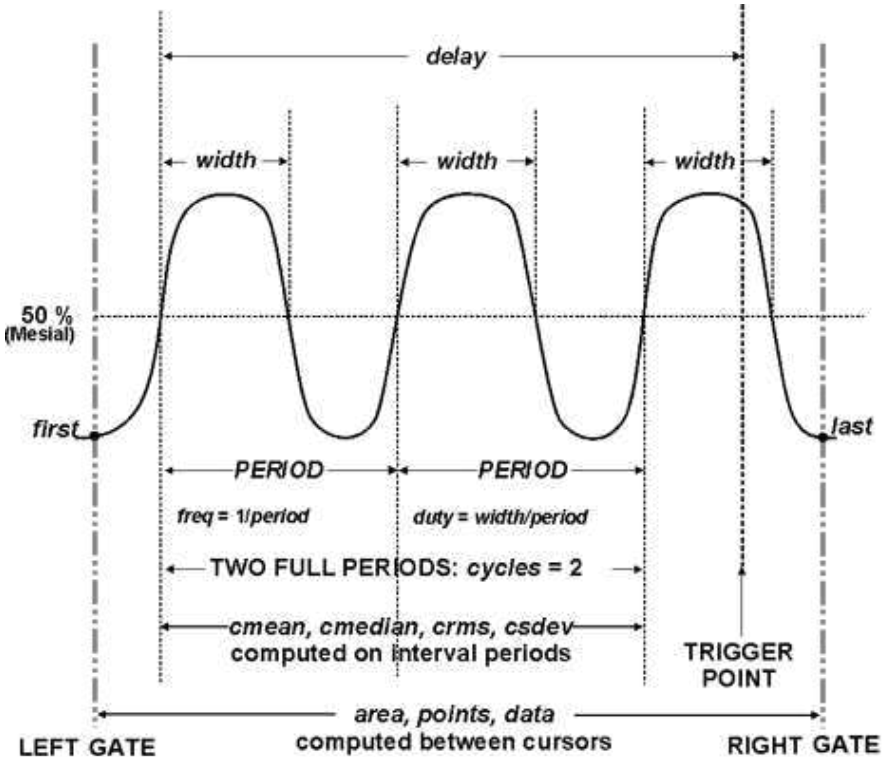
$$\frac{1}{Mf}\sum_{i=1}^{Mf}(Tf_i^{10} - Tf_i^{90})$$

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.

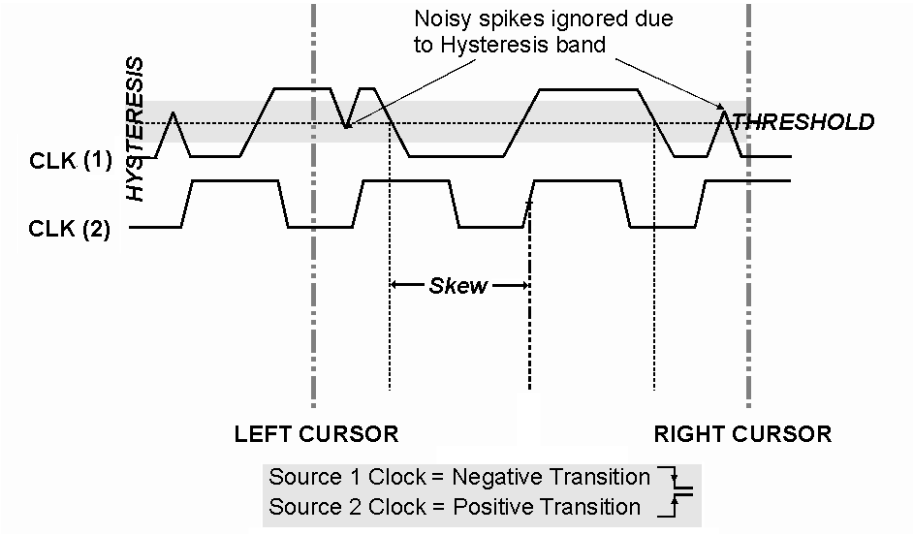
Time-parameter estimation depends on the number of cycles included within the observation window. If the number of cycles is not an integer, parameter measurements such as rms or mean will be biased. However, only the last value is actually displayed, the mean being available when statistics are enabled. 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 datasheets on the Teledyne LeCroy website 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.

Single vs. Dual Operator Functions

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

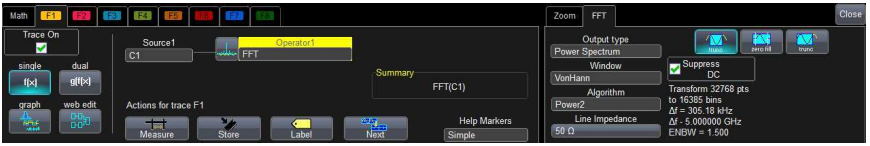
Dual functions chain two operations to arrive at a single result. This saves you the effort of having to chain two separate math functions. As with single functions, the number of sources required will vary based on the operation. You may need only one source for Operator1, but two for Operator2 (the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

Setting Up Math Functions

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

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**.

   TIP: 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.

   TIP: 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.
5. The choice of operator drives the number of **Source** fields you will see displayed. Make a selection in each field.

   A **Summary** of the function you are building appears on the dialog. Refer to this to be sure your sources are in the proper order to yield the function you want (e.g., C1-C2 vs. C2-C1).

6. If the operator you've selected has any other configurable settings, you'll see a right-hand dialog of the same name as the operator. Touch the tab to open the dialog and make any further settings. These are explained on the dialog.

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

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

Enable/Disable Math Function

Once a math function has been created and saved, just use the main Math dialog to quickly enable/disable it.

Touch the front panel **Math button**, or from the menu bar choose **Math > Math Setup**, then check the **On** box next to each function you wish to display.

Touch **Reset All** to erase all functions from their locations.

Touch **Clear Sweeps** to restart the counter on cumulative functions (like Average).

Graphing

The **Graph button** on the Function (F*x*) dialogs allows you to plot the results of an applied measurement parameter using histogram, track, or trend. Choose the source, the measurement parameter, and the type of plot to draw. See Histogramming and Track vs. 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 right-hand dialogs after the plot is selected.

Adjust 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 right-hand dialog,** or on one of the trace setup dialogs. 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, however, create a separate zoom trace from a memory or function trace by drawing a selection box around a portion of the waveform. 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.

List of Standard Operators

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

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

Operator	Definition
Absolute	For every point in the waveform the distance away from zero is calculated. For values greater than zero this is the same as the value. For values less than zero, the magnitude of this value without regard to its sign is used.
Average	Calculates either a summed or continuous average of a selected number of sweeps. See Averaging Waveforms. The maximum number of sweeps is determined by the oscilloscope model and memory. See the specifications at teledynelecroy.com.
Copy	Copies waveform in its unprocessed state to the first available memory location.

Operator	Definition
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 1 Mpts. Also allows FFT Averaging through use of a second math operator. See FFT.
Floor	Calculates the lowest vertical values of a waveform at each horizontal value for a specified number of sweeps.
Integral	Calculates the linearly rescaled integral (with multiplier and adder) of a waveform input starting from the left edge of the screen using the formula: *(current sample value + next sample value) * (horizontal sample interval)* Each calculated area is summed with the previous sum of areas. The multiplier and adder are applied before the integration function.
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.
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

Operator	Definition
	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 persistence histogram based on the displayed pixels of a waveform falling within a user defined vertical or horizontal box (slice).
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.
Ptrace range	Generates a waveform with a width derived from the population range of a persistence map.
Ptrace sigma	Generates a waveform with a width derived from the sigma (sum) of a persistence map.
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	Performs10 -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.

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

Interpolate Function

Linear interpolation, which inserts a straight line between sample points, 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 waveshapes, especially when the sampling rate is 3 to 5 times the system bandwidth. The instrument also gives you a choice of Cubic interpolation. For each method, you can select a factor from 2 to 50 points by which to interpolate (upsample).

1. Follow the usual steps to set up a math function, selecting **Interpolate** from the **Filter** submenu.
2. Open the **Interpolate** right-hand dialog.
3. Touch **Algorithm** a nd select an interpolation type.
4. Touch **Upsample by** and enter the factor by which to increase sampling.

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.

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 right-hand dialog, optionally **Reset Count** or **Change BatchSize**.

Rescaling 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.

Set Up Rescaling

1. Follow the usual steps to set up a math function, selecting **Rescale** from the **Functions** submenu.
2. Touch the **Rescale** right-hand dialog tab.
3. To apply a multiplication factor:
   - Check the **First multiply by:** box and enter a value for *a*, the multiplication factor.
   - Touch **then add:** and enter a value for *b*, the additive constant.
4. To change the output unit of measure from that of the source waveform:
   - Check **Override units**.
   - In **Output** enter the abbreviation for the 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: "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		

Enhanced Resolution

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

Use ERes:

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

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

Set Up Enhanced Resolution (ERes)

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

To apply ERes as a Math function:

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

How the Instrument Enhances Resolution

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

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

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

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

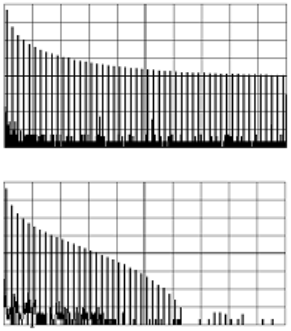
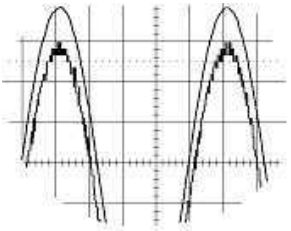
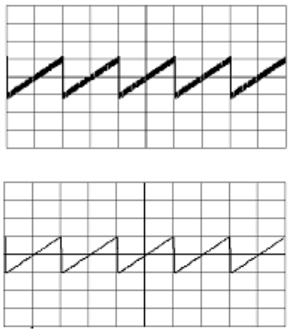
The improvement in SNR corresponds to the improvement in resolution if the noise in the signal is white (evenly distributed across the frequency spectrum). If the noise power is biased towards high frequencies, the SNR improvement will be better than the resolution improvement.

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

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

Example ERes Applications

The following examples illustrate how you might use the ERes filter.

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

NOTE: While enhanced resolution can only improve the resolution of a trace, it cannot improve the accuracy or linearity of the original quantization. The pass-band causes signal attenuation for signals near the cut-off frequency. The highest frequencies passed may be slightly attenuated. Perform the filtering on finite record lengths. Data is lost at the start and end of the waveform and the trace ends up slightly shorter after

filtering. The number of samples lost is exactly equal to the length of the impulse response of the filter used: between 2 and 117 samples. Normally this loss (just 0.2 % of a 50,000 point trace) is not noticed. However, you might filter a record so short that no data is output. In that case, however, the instrument would not allow you to use the ERes feature.

Averaging Waveforms

Set Up Averaging

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

To apply Continuous or Summed Averaging as a Math function:

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

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

NOTE: Continuous Averaging may be set up as Pre-Processing on the Channel dialog or as a Math function.

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

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

new average = (new data + weight * old average)/(weight + 1)

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

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.

FFT

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

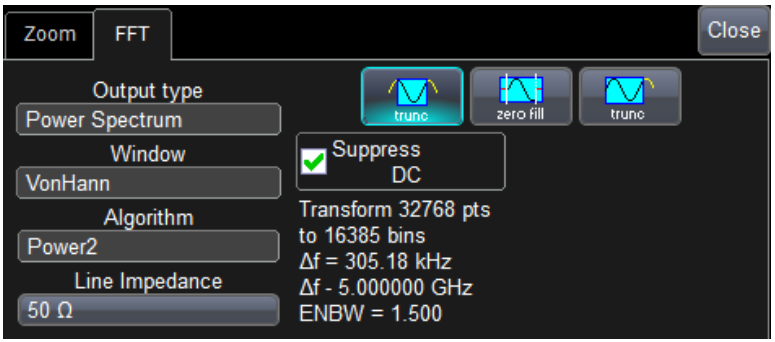
If sampling is done at a rate fast enough to faithfully approximate the original waveform (usually five times the highest frequency component in the signal), the resulting discrete data series will uniquely describe the analog signal. This is of particular value when dealing with transient signals, which 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** right-hand dialog.



3. Choose an **Output type**.
4. Optionally, choose a weighting **Window** (see below).
5. Depending on your **Output Type** selection, also make selections for :

- **Group Delay Shift**
- **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.

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

Memory

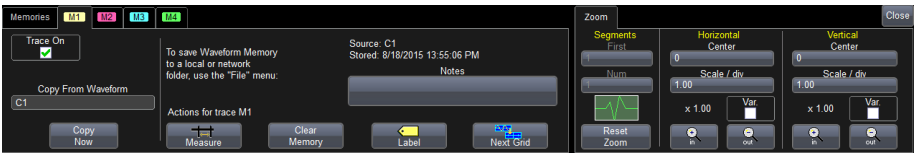
The instrument is equipped with four internal memory slots (M*x*) to which you can copy any waveform that is active on the grid. This is a convenient way to store an acquisition for later viewing and analysis.

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

Save Waveform to Memory

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

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



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

Save 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.

Restore Memory

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

Analysis

Most Teledyne LeCroy 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 waveform trace. The standard Analysis menu tools go further to help you understand the behavior of waveforms.

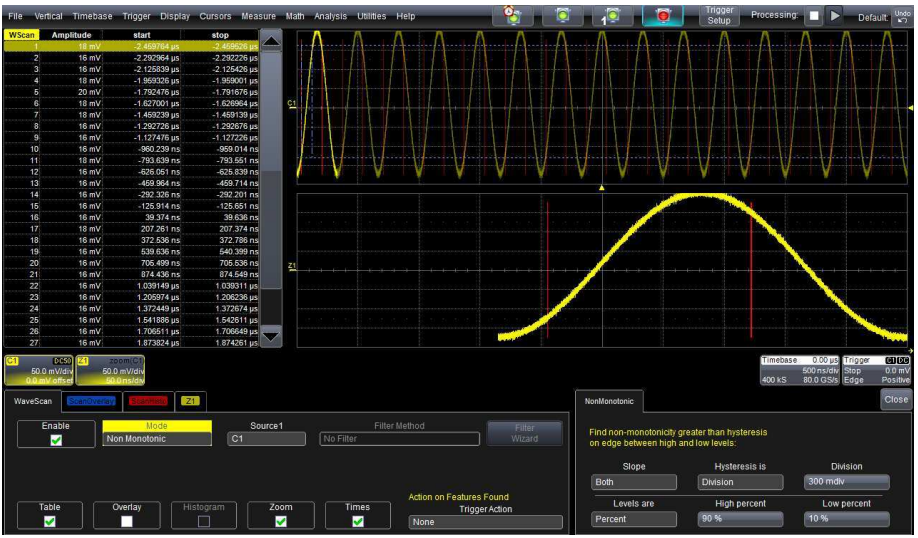
WaveScan searches a single acquisition for events that meet specific criteria, enabling you to zoom in on anomalies in the waveform, or scans multiple acquisitions with allowable trigger actions when conditions are met. It can also be used to filter measurements.

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

Optional software packages may be purchased for specialized uses, such as power analysis. In most cases, these options are added to the Analysis menu.

WaveScan

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



WaveScan window with different scan "views" turned on.

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

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

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. Non-monotonic Mode 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. You can choose to display views simultaneously or visit them sequentially. Just check the boxes at the bottom of the WaveScan dialog for those views you wish to display. Uncheck the box to turn off the view.

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

You'll find additional controls for manipulating Scan Overlay and Zoom 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 in the trace 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 plots the location of captured events in a new trace. Colored overlays provide quick reference to sections of the waveform.

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 Controls for an explanation of the remainder of the controls found on this dialog.

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

Setting Up WaveScan

This procedure explains how to set up WaveScan to search an acquisition for events of interest. Set up your source channel and triggers before setting up the scan.

1. Press the front panel **Stop button** to stop acquisition.
2. Choose **Analysis > WaveScan**.
3. Check **Enable**.
4. Choose the **Source** waveform.
5. Choose the Scan Mode and enter values for any additional settings that appear at the right of the dialog based on your selection.
6. If you're using Measurement Mode, set up the filter in one of the following ways:
   - Touch **Filter** and choose an operator, then enter the **Filter Limit**.
   - Touch **Filter Wizard** and choose one of the pre-set filters. The Filter and Filter Limit are automatically set based on your selection.
7. Select each Scan View in which you wish to display results by checking the box at the bottom of the dialog. Each view selected is displayed simultaneously.
8. If you're using Scan Overlay view, on the Scan Overlay dialog **Clear Sweeps**. If desired, set up the Persistence display.
9. Optionally, choose an **Action** to trigger when an event that meets your scan criteria is found.
10. Restart acquisition.
11. When using the Zoom view, use the Z*x* tab to adjust the zoom.

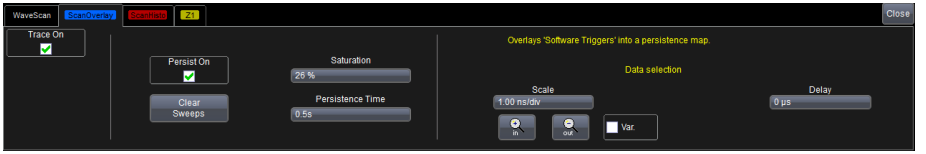
Adding Persistence to ScanOverlay

To apply monochromatic persistence to a Scan Overlay:

1. Select **Scan Overlay** when setting up the wave scan, then open the **ScanOverlay 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 effectively "zoom" in or out: touch the **In/Out buttons**, or touch **Scale** and **Delay** and enter new values.

Check **Var.** to adjust values in finer steps than the default 1, 2, 5.



Scan Histogram

Scan Histogram is an additional "view" that generates a histogram to give you a statistical view of edges that meet your search criteria. Enter the parameters as you would to set up any histogram:

1. Total **#Values** in the histogram buffer
2. **#Bins** (bars) in the histogram
3. **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.

Pass/Fail Testing

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

There are two principal Pass/Fail testing methods:

- Mask testing, where sampled values are tested to see if they fall within a pre-defined area of the grid (the "mask")
- Parameter comparison, where a measurement parameter (P*x*) is compared to a pre-defined value (Param compare) or to another measurement parameter (Dual param compare)

You can set up to qualifiers (Q*x*) using either method, which can then be enabled together or separately during a single test.

Set Up Mask Qualifiers

Testing against a mask is particularly useful for comparing newly acquired signals to a previously acquired "golden standard" waveform.

A mask defines an area of the grid against which a source Channel, Zoom,

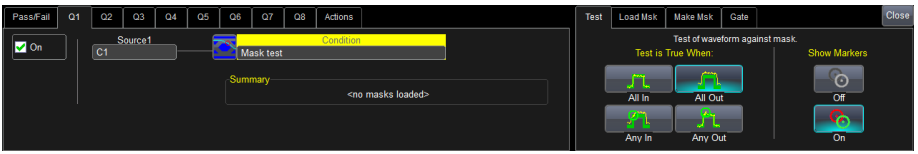
or Math trace is compared. Test conditions are associated with the mask, defining how the waveform is to be compared to the masked area (e.g., some/all values fall within, some/all values fall outside), and a Pass or Fail result is returned indicating the condition was found to be true or false.

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

Access Pass/Fail Test Dialogs

Choose **Analysis > Pass/Fail** to display the **Pass/Fail** dialog. Touch the **Q*x* button** where you want to set up the qualifier. From the pop-up menu, select **Mask test**.

The Q*x* dialog opens with the Mask test condition selected and the **Test**, **Load Mask**, **Make Mask**, and **Gate** right-hand dialogs 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. Touch the **Make Mask tab** to display the dialog.
2. If desired, enter a new **Destination File Name** and path, or touch Browse and select a previous file to overwrite. The file name should end with the **.msk** extension.
3. Touch the **Ver Delta** and **Hor Delta** fields and enter boundary values using the pop-up numeric keypad or the front panel Adjust knob.
4. Touch **Make from Trace**.

Load Mask

Use this procedure in lieu of Make Mask if you have a pre-defined mask file, or wish to recall a mask you previously created and saved.

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

Set Gates

Set gates to limit the portion of the waveform that is compared to the mask.

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

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

Define Test

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

Removing a Mask from the Display

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

   OR

   Open the **Q*x*** dialog where a particular mask is set, and on the right-hand **Load Mask** dialog, touch **Delete**.

Set Up Param(eter) Compare Qualifiers

Pass/Fail qualifiers (**Q*x***) can be configured to compare different parameter measurements to each other or to a user-defined limit (or statistical range). The parameters themselves (**P*x***) are set up on the Measure dialog, then selected here for comparison.

Comparing a Measurement to a Limit

This method takes the result of a selected measurement parameter and compares it to a user-defined value or range of values.

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 dialog, touch **Compare Values** and select **All** or **Any** from the pop-up.

   By selecting **All**, the condition is true only if every waveform falls within the limit. Selecting **Any** makes the condition true if just one waveform falls within the limit.

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

When using Dual Param(eter) Compare, two measurements, rather than two waveforms, are compared.

1. Choose **Analysis > Pass/Fail** to display the **Pass/Fail** dialog.
2. Touch the **Qx 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 dialog, touch **Compare values** and select **All** or **Any** from the pop-up.

   By selecting **All**, the test are true only if every waveform falls within the set limit. Selecting **Any** makes the test true if just one waveform falls within the limit.

6. Touch **Condition** and select the math operator that completes the condition. The Summary field shows the completed condition that is represented by this qualifier.

Define Pass/Fail Tests

The Qualifiers define the conditions of a Pass/Fail test 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 open the **Aux Output tab** and choose to **Use Auxilliary Output For Pass/Fail**.
     - Capture the screen and process it according to your **Hardcopy** selection (e.g., send to printer)
     - Create a **LabNotebook** Notebook Entry

   Use the **Clear All** button to clear all the action checkboxes, or **Force Actions Once** to apply them once regardless of the test results.

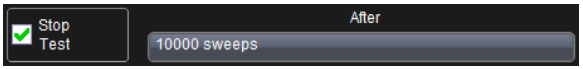
Delay Test

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**.



Utilities

Utilities Settings

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

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

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

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

NOTE: Hardcopy Setup controls the behavior of the Print function. The selected print output device or application is displayed to the right of the **HardCopy Setup** button for convenience.

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.

System Status

The Utilities Status dialog displays information about your instrument including model number, serial number, firmware version, and installed hardware and software options.

Choose **Utilities > Utilities Setup** from the menu bar, then touch the **Status tab**.

OR

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

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

Remote Control Settings

The Remote dialog contains settings to configure remote control of the instrument 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. 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.

Set Up Remote Control

NOTE: Full remote control setup requires the installation and configuration of software on the controller, as well. These steps represent only what is done on the instrument. See the Remote Control manual for your model for an explantion of the complete process.

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 controller privileges to specific network clients, touch **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**.

Hardcopy (Print) Settings

Hardcopy settings control how the **Print** function 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 to a hardcopy printer
- "Print" to a file that can be saved to an internal or external drive
- Send via e-mail
- Copy to the Windows clipboard to be pasted elsewhere

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

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

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

SEND TO PRINTER

Add Printer

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

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

Print Setup

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

PRINT TO FILE

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

1. On the Hardcopy dialog, choose **File**.
2. Choose the output **File Format**.
3. Enter a **File Name**. This will form the basis of all print filenames, until you change it.

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

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

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

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

COPY TO CLIPBOARD

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

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

Send to E-Mail

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

1. On the Hardcopy dialog, choose **E-Mail**.
2. Choose the output **File Format**.
3. If you wish to be able to include messages with the files as they are sent, check **Prompt for message to send with mail**.
4. Optionally, choose a color scheme and hardcopy (print) area.
5. To go on and set up the e-mail connection, touch **Configure E-Mail Server and recipient**.

Choose Print Color Scheme

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

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

NOTE: The colors used to represent channels in Standard and Print schemes are configured on the Preferences Colors dialog.

Set Print Area

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

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

Auxiliary Output Settings

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

AUXILIARY OUTPUT

Square outputs a square wave. Enter the desired Amplitude and Frequency.

Trigger Enabled sends a pulse when the trigger is ready (Ready indicator lit), but not necessarily fired. It can be used as a gating function to trigger another instrument. Enter the desired pulse **Amplitude**.

Trigger Out sends a pulse when the trigger fires (Trig'd Indicator lit). Enter the desired pulse **Amplitude**.

Pass/Fail generates a pulse when Pass/Fail testing is active and conditions are met. With this selection, a **Pulse Duration** data entry control appears. Provide a value within your instrument's specified range, which varies by model. Refer to datasheet specifications at teledynelecroy.com.

Reference **DC Level**. Enter the desired **Amplitude**.

Fast Edge provides a fast edge signal that can be used to deskew multiple channels.

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. With this selection, also 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**.

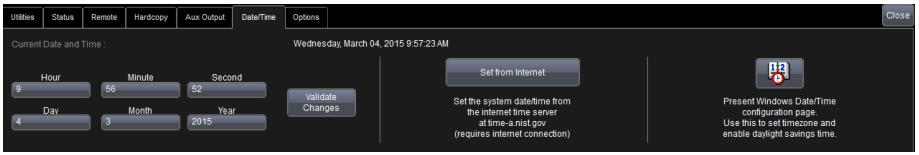
Off disables calibration output.

Date/Time Settings

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

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

To access the Date/Time dialog, choose **Utilities > Utilities Setup** from the menu bar, then touch the **Date/Time tab**.



MANUAL METHOD

Enter the **Hour**, **Minute**, **Second**, **Day**, **Month**, and **Year**, then touch the **Validate Changes** button.

INTERNET METHOD

This method uses the Simple Network Time Protocol (SNTP) to read the time from time-a.nist.gov. The 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

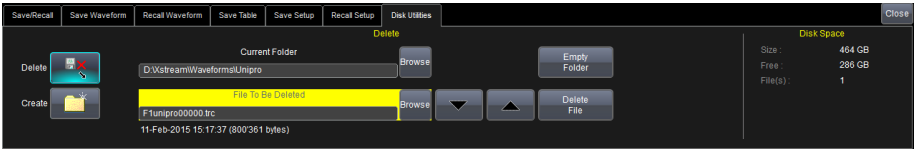
The **Options** dialog is used to add or remove software options. This dialog also displays the **ScopeID** and **Serial #**. See Adding an Option Key for instructions on using this dialog.

Disk Utilities

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

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

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



Delete a Single File

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

Delete All Files in a Folder

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

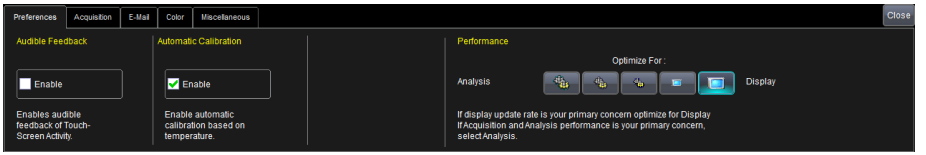
Create a New Folder

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

System Preferences

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

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



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

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

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

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, E-Mail, Color, and Miscellaneous settings.

Acquisition Preferences

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

Offset Setting constant in:

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

Delay Setting constant in:

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

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

Check **Reset trigger counter before starting a new acquisition** to clear the trigger counter each time a new acquisition command is sent. It is only available when trigger Holdoff is set.

HorScale (T/Div) always canonic:

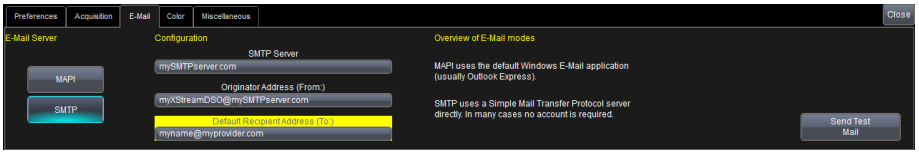
- **Off**
- **On** retains 1,2,5 step scaling when in Maximum Memory mode.

Digitizer Mode:

- **Off**
- **On** time-aligns sample points for all source channels.

E-Mail Preferences

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



1. Under **E-Mail Server**, select the protocol used by your network:
   - **MAPI** (Messaging Application Programming Interface) is the Microsoft interface specification that allows different messaging and workgroup applications (including e-mail, voice mail, and fax) to work through a single client, such as the Exchange client. MAPI uses the default Windows e-mail application.
   - **SMTP** (Simple Mail Transfer Protocol) is a TCP/IP protocol for sending messages from one computer to another through a network. This protocol is used on the Internet to route e-mail. In many cases no account is needed.
2. Under **Configuration**:
   - **If you chose SMTP**, touch **SMTP Server** and enter the network address of your mail server.
   - Touch **Originator Address (From:)** and enter the instrument's e-mail address.
   - Touch **Default Recipient Address (To:)** and enter the recipient's e-mail address.
3. Touch **Send Test Mail** to send a confirmation message to ensure proper e-mail configuration.

Color Preferences

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 the Hardcopy dialog in **Utilities > Utilities Setup...**. 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 Preferences

These other Preference settings are located on the **Miscellaneous** dialog.

To add the Teledyne LeCroy logo to print output, check **Print Teledyne LeCroy Logo When Printing Grid Area Only**. This identifies the instrument as the source of the image.

You can adjust zoom behavior as follows:

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

Serial Decode Annotation Position: If you have Serial Trigger or Decode options installed on your 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.

Save/Recall

The **File menu** allows you to save or retrieve waveform files, measurement table data, and instrument setup panels. There are also Disk Utilities for arranging the file/folder structure on your instrument's hard drive.

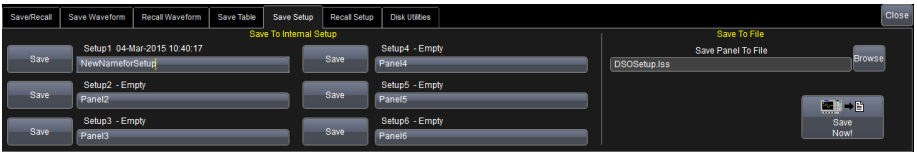
Access these functions by choosing any of the Save or Recall options from the File menu. The dialog contains a tab for each file function.

Save Setups

Save Setups allows you to quickly save up-to-six panel settings to internal storage, while Recall Setups restores them with a touch. Setups are saved to the D:\Internal Setups folder.

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

Choose **File > Save Setup...** from the menu bar or click the **Save Instrument Setup** button on the main Save/Recall dialog.



Save Setup to Memory

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

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

Save Setup to File

1. In **Save Panel to File**, touch **Browse** and navigate to the desired folder.

   TIP: If the instrument is networked, you can touch Save Panel to File and enter the full Windows network address of another location in which to save the file. The instrument must have access to this directory.

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

Recall Setups

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

Choose **File > Recall Setup...** from the menu bar or click the **Recall Instrument Setup** button on the main Save/Recall dialog.

Recall Setup from Memory

Touch one of the six **Recall** buttons under **Recall From Internal Setup...**.

NOTE: If a setup has been stored to a location, it is labeled with the save date/time. Otherwise, the slot is labeled **Empty**.

Recall Setup from File

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

Save Waveforms

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

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

By default, trace files are saved to the D:\...\Waveforms folder on the instrument hard drive, although you can choose another location, such as a USB drive. The file name is autogenerated from the <source trace><trace title><number in sequence> (e.g., C1test000001).

Choose **File > Save Waveform** from the menu bar or click the **Save Waveform** button on the main Save/Recall dialog.

Save Waveform To Memory

1. Touch **Memory**.

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

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

Save Waveform To File

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

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

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

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

   - **ASCII** text file (.txt extension).
   - **MATLAB** text file (.dat extension).
   - **Excel** text file (.csv extension).
   - **MathCad** text file (.prn extension).
   - **Audio** .wav file.
   - **WaveML**, Teledyne LeCroy's proprietary .xml format used to save persistence maps, eye diagrams, histograms, and digital traces. This option will only be activated if the source waveform/plot is of a type to require it.
5. Depending on your file format selection, you may also need to specify a **SubFormat:**

- **Word** (Binary) represents samples in the output file with 16 bits. Always use this options unless Byte mode is "pre."
- **Byte** (Binary) represents samples in the output file with 8 bits. This option can result in a loss of output file resolution.
- **Auto** (Binary) looks at the data and automatically selects either Word or Byte subformat.
- **Amplitude only** (Text) includes amplitude data for each sample, but not time data.
- **Time and Amplitude** (Text) includes both types of data for each sample.
- **With Header** (Text) includes a file header with scaling information.

6. If you selected **ASCII** format, also touch **Delimiter** and select a delimiter character from the pop-up menu.
7. In Save Files in Directory, touch **Browse** and navigate to the desired location. Touch **OK**.

   TIP: If the instrument is networked, you can touch on Save Files in Directory and enter the full Windows network address of another location in which to save the file. The instrument must have access to this directory.

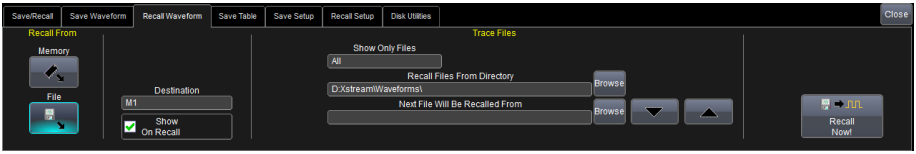
8. On the Save Waveform dialog, touch **Save Now!**

Recall Waveforms

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

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

Choose **File > Recall Waveform** from the menu bar or click the **Recall Waveform** button on the main Save/Recall dialog.



Recall Waveform From Memory

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

Recall Waveform From File

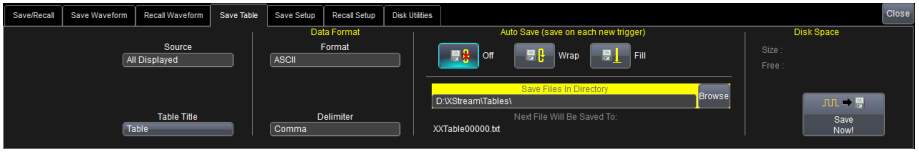
1. Touch **File**.
2. Touch **Recall files from directory** and enter the path to the waveform folder, or touch **Browse** and navigate to the folder.
3. Use the **Up /Down Arrows** to cycle through the available files until the desired file is selected.

   Optionally, touch **Show only files** to apply a search filter (**channels**, **math functions**, or **memory**) to the list of available files.
4. Mark **Show on Recall** to display the trace on the grid.
5. Touch **Recall Now!**

Save Table Data

The Save Table function saves tabular measurement data displayed on screen to an Excel or ASCII file. By default, files are saved in D:\...\Tables, although you can choose another location.

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



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

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

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

   TIP: If the instrument is networked, you can touch Save Files in Directory and enter the full Windows network address of another location in which to save the file. The instrument must have access to this directory.

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

LabNotebook

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

Notebook Entries can then be output to a hardcopy report format—.pdf, .rtf, or .html—and printed or e-mailed. You can also design and upload your own report layout if you prefer not to use the default.

Entries can be collected into separate Notebooks by project or user, especially useful if the instrument is shared. Similarly, you can customize the folder structure into which Notebooks are stored to facilitate backup and sharing.

Notebook Entries are stored in an internal database and are available to be recalled to the touch screen at any time. A keyword filter makes it easy to find and recall a specific Notebook Entry. Besides storing the waveform data, LabNotebook also stores your panel setups and parameter measurements. Back up this database to external media for indefinite storage of waveform data.

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

Create Notebook Entry

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

1. Choose **File > LabNotebook** to open the LabNotebook dialog.
2. **Select the notebook** to which to add the entry from the My Notebooks list.

   New entries are added to whichever notebook was last selected, or to the default MyNotebook.zip if you have only one.
3. Touch **Create**.
4. Optionally, **Enter Report Title** and **Description**.

   The default title is the date and time stamp.

   NOTE: By default, you will be prompted to title and annotate notebook

entries as they are created. You can configure LabNotebook preferences so that these steps are skipped in order to streamline the creation process. To update entries at a later time, select the entry from the list of Notebook Entries, then open the tab of the same name that appears behind the LabNotebook dialog.

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

LabNotebook Drawing Toolbar

The basic Notebook Entry is a screen capture of the display as it was at the time of entry, along with the setup and waveform data. When an entry is first captured, it is immediately displayed in the Drawing window for you to annotate.

TIP: This default setting can be changed on the LabNotebook Preferences dialog.

Markup tools are available from the toolbar along the top of the window.

To use any tool, touch the icon, then touch the point on the image where you wish to draw or add text. From left to right, the tools are:

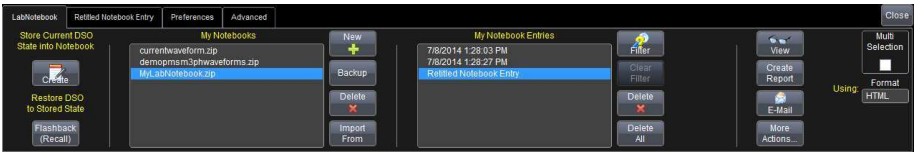
Tool	Function
Pencil	Draw in freehand. Maintain contact with the screen to make a continuous mark. Once you release, you can touch-and-drag the object to any point on the image.
Circle	Draw a circle around a waveform feature that you want to emphasize. Touch-and-drag across the diameter of the circle. 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.

Tool	Function
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.

Manage Notebook Entries

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

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



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

Select Notebook Entries

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

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

   OR

   To select multiple entries, first check Multi-selection, then Select All or

scroll the list touching Select as a desired entry is highlighted. A white arrow appears next to each selected entry.

3. To remove selections from the list, highlight them again and touch **Clear**.

Filter Notebook Entries

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

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

View Notebook Entries

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

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

Edit Notebook Entries

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

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

Email Notebook Entries

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

If you have not yet configured email, or if you wish to change the recipient

address before sending, open the LabNotebook Preferences tab, then touch the **Configure E-Mail** button.

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

Print Notebook Entries

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

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

Delete Notebook Entries

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

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

Create Report

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

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

   TIP: Touch **Open Explorer Here** and use the Windows Explorer to create a new folder. After closing the Explorer, touch the **Refresh** button to display the folder in the Create Report window.

6. Enter a **File name** for the report and click **OK**.

Manage Notebooks

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

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

Create New Notebook

1. Choose **File > LabNotebook**.
2. Touch the **New** button next to the My Notebooks list.
3. Enter a **File Name** for the new Notebook (optionally, choose a new storage folder, as well). Touch **OK**.

The new notebook now appears in the My Notebooks list. New Notebook Entries will be added to this Notebook whenever it is selected.

Back Up Notebook

1. Choose **File > LabNotebook**.
2. Select the notebook from the **My Notebooks** list and touch **Backup**.
3. Optionally, enter a new **File Name** or choose a new storage **Folder**.

   NOTE: The default is the notebook name with *.bak.zip appended to it.
4. Choose to **Backup to Removable Disk** (this option is active when a USB drive is connected) or **Backup to Folder** on hard drive.

Import Notebook

Archived notebooks can be imported into the My Notebooks list.

1. Choose **File > LabNotebook**.
2. Touch the **Import** button.
3. Navigate to the desired archive and select it. Touch **OK**.

Delete Notebook

1. Choose **File > LabNotebook**.
2. Select the notebook from the My Notebooks list.
3. Touch the **Delete** button next to the My Notebooks list.

Print to Notebook Entry

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

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

Flashback Recall

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

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

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

- **Persistence data** (although it is saved in with the Notebook Entry and appears on reports).
- **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 Recall is entered. When Flashback is used, they lose their history and show instead only the results from the stored waveforms, not including any data taken from interim acquisitions.

Customize Reports

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

Change Directories

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

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

Change Report Template

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

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

Change Logo

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

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

LabNotebook Preferences

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

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

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

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

Use Print Colors outputs waveforms on a white background. The print

colors used for each trace are set in **Utilities > Preferences Setup > Colors**. This option helps save ink/toner when printing.

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

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

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

Maintenance

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.

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.

WARNING. 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 to the back of the instrument, then 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.

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.

Restart/Reboot Instrument

To restart the X-Stream application, choose **File > Exit** then touch the **Start DSO** desktop shortcut.

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

To reboot the instrument, which includes restarting the OS:

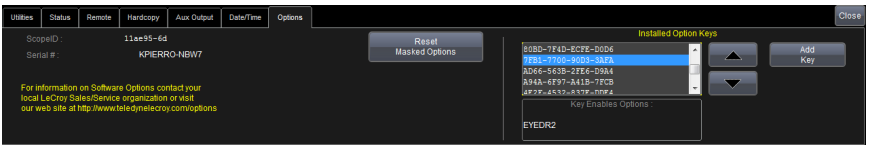
1. Shut down by choosing **File > Shutdown** from within the X-Stream application, or using the Windows Start Menu Shutdown command.
2. Wait 10 seconds, then press the **Power button** on the front of the instrument.

Adding an Option Key

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 enables the new functionality.

To install the key and activate the software:

1. From the menu bar, choose **Utillities > Utilities Setup**, then touch the **Options tab**.



2. Touch **Add Key**. The Virtual Keyboard appears onscreen.
3. Use the VIrtual Keyboard to type the Key Code in the **Enter Option Key** field, then touch **O.K.** on the keyboard to enter the information.

   The Key Code is added to the list of Installed Option Keys. You can use the Up/Down buttons to scroll the list. The software option that each key activates is displayed below the list.
4. Restart the X-Stream application: choose **File > Shutdown**, then double-click the **Start DSO** icon on the desktop.

X-Stream Firmware Update

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

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** to return to the X-Stream Setup wizard.
10. When installation is complete, choose **Reboot now** and click **Finish**.

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

System Recovery from Hard Drive

To correct instabilities resulting from the instrument's Windows OS, Teledyne LeCroy provides a recovery application and a backup image in an extra partition on the instrument's hard drive.

Generally, models running on 64-bit Windows 7 platorms will have Acronis True Image Home installed.

You must reactivate Windows after recovery is complete, either by Internet connection to Microsoft's website or by telephone. Have your Windows Product Key number handy for Widows reactivation.

Before You Start

- If you intend to reactivate Windows through the Internet, connect the instrument to your network.
- Connect a keyboard and mouse to the instrument.
- Note the Windows Product Key number and serial number, usually listed on a sticker on the back of the instrument.
- Power down the instrument.

Determine the Recovery Product

1. Power on the instrument.
2. During the startup process, when you see "Starting Acronis Loader...", press the **F11 key** until the Acronis logo appears momentarily. The Acronis window is displayed.

   NOTE: Do not press F11 before you see "Starting Acronis Loader..." If a boot menu dialog appears, press Cancel or Esc.

   Check the Acronis window to determine which product you are using. Follow Using True Image Home or Using Echo Workstation accordingly.

   Or

   If you do NOT see "Starting Acronis Loader..." during your boot sequence, press the **F4 key** when the LeCroy logo appears and follow Using Phoenix cME.

Using True Image Home

1. Select **Acronis True Image Home (Full Version)**.
2. On the Acronis True Image Home page, under options for Recover, select **My Disks**. The Recovery Wizard opens.
3. On the Recovery Wizard, under Archive Selection, **select the disk**

archive with a create date, then click **Next**.

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

CAUTION. Recovery takes 4 - 15 minutes. Do not power down or otherwise interrupt the recovery while in process.

9. When recovery is complete, you will see "Recover operation succeeded." Click **OK**.
10. Click the **close button** to exit the Acronis window. The instrument will restart and begin installing the required software. Continue to reinstall the X-Stream application.

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

Using Echo Workstation

1. On the Pick a Task page, select **Recovery**. The Restore Data Wizard opens.
2. Click **Next** on the Welcome page.
3. On Backup Archive Selection, choose **Acronis Secure Zone** (this is where the recovery data is located on the instrument). Click **Next**.
4. On Restoration Type Selection, select **Restore disks or partitions**, then click **Next**.
5. On Partition or Disk to Restore, select **SYSTEM (C:)** as the source, then click **Next**.
6. On Restored Partition Location, select **SYSTEM (C:)**, then click **Next**.
7. On Restored Partition Type, select **Active**, then click **Next**.

Note: If asked if you want to buy Acronis products, check "Do not show this message again" and click OK.

8. On Restored Partition Size, simply click **Next**.
9. On Next Selection, select **No, I do not** (want to restore another

partition), then click **Next**.

10. On Restoration Options, click **Next** to use the default selections.
11. A summary window indicates that Acronis True Image is ready to proceed with the recovery of the C: partition. Click **Proceed** to start the recovery.

CAUTION. Recovery takes 4 - 15 minutes. Do not power down or otherwise interrupt the recovery while in process.

12. Once the recovery is complete, an Information dialog indicates that the Data was successfully restored. Click **OK**.
13. From the Acronis menu bar, choose **Operations > Exit**. The oscilloscope will restart and begin installing the required software.

Using Phoenix cME

1. When the cME console End User License Agreement is shown, read the agreement and click **Accept**.
2. On the Phoenix cME Console main page, click the link "**Click here to start recover**".

   The recovery starts shortly after the FirstWave flash page, and the FirstWare Progress screen is displayed. No further selections are required.The recovery takes about 10 minutes.

   NOTE: The screen occassionally goes blank for prolonged periods. This is normal and not an indication of malfunctioning.

3. After recovery the X-Stream DSO Setup Wizard appears. Go on to re-install the oscilloscope application.

Reinstall the X-Stream Application

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

   NOTE: If you see a message that Windows can't verify the publisher of the driver software, choose "Install this driver software anyway" and click Install.

4. When the X-Stream installation is complete, choose to **Reboot now** and click **Finish**.
5. When prompted, enter your Windows Product Key number to re-activate Windows.

Rename Instrument

In order to apply option keys to the instrument in the future, restore the correct name (serial number).

1. Minimize the oscilloscope application window.
2. From the Windows **Start Menu**, choose **Control Panel > System**.
3. Scroll down to **Computer name, domain, and workgroup settings**, then click **Change Settings**.
4. On the System Properties dialog, click **Change** and enter the oscilloscope serial number in **Computer name**.
5. Click **OK** twice to return to the desktop.

Reactivate the F11 Startup Utility

Phoenix and Echo Workstation users can skip this procedure and go on to update software to current levels.

In order for the system recovery wizard to be accessed again from the boot menu, reactivate the F11 startup utility. It is critical to complete these steps after restarting.

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

   NOTE: The process may take several minutes.

Technical Support

Phone

Registered users can contact their local Teledyne LeCroy service center at the number listed on our website. For a complete list of sales and service center by country, visit:

teledynelecroy.com/support/contact

Web

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

The Datasheet published on the product page contains the detailed product specifications.

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

teledynelecroy.com/support/techhelp

Returning a Product for Service

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

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

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

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

Contact Us

Our regional service centers are:

World Wide Corporate Office

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
support@teledynelecroy.com

US Protocol Solutions Group

Teledyne LeCroy
3385 Scott Boulevard
Santa Clara, CA, 95054, USA
teledynelecroy.com
Sales and Service:
Ph: 800-909-7211 / 408-727-6600
FAX: 408-727-0800
protocolsales@teledynelecroy.com
Support:
Ph: 800-909-7112 / 408-653-1260
psgsupport@teledynelecroy.com

European Headquarters

Teledyne LeCroy SA
4, Rue Moïse Marcinhes
Case postale 341
1217 Meyrin 1
Geneva, Switzerland
teledynelecroy.com/europe
Ph: + 41 22 719 2111
FAX: + 41 22 719 2230
contact.sa@teledynelecroy.com

China

LeCroy Corporation Beijing
Rm. 2001, Unit A, Horizon Plaza
No. 6, Zhichun Road, Haidian Dist.
Beijing 100088, China
www.lecroy.com.cn
Sales:
Ph: 86-10-82800318/0319/0320
FAX:86-10-82800316
Marketing.China@teledynelecroy.com
Service:
Rm. 2002
Ph: 86-10-82800245
Service.China@teledynelecroy.com

Korea

Teledyne LeCroy Korea
10th fl. 333 Yeongdong-daero
Gangnam-gu
Seoul 135-280, Korea
teledynelecroy.com/korea
Ph: ++ 82 2 3452 0400
FAX: ++ 82 2 3452 0490

Japan

Teledyne LeCroy Japan
3F, Houbunshafuchu Bldg.
3-11-5, Midori-cho, Fuchu-Shi
Tokyo 183-0006, Japan
teledynelecroy.com/japan
Ph: + 81-42-402-9400
FAX: + 81-42-402-9586

For a complete list of offices by country, including our sales & distribution partners, visit:

teledynelecroy.com/support/contact

Certifications

Teledyne LeCroy certifies compliance to the following standards as of the time of publication. As standards evolve, these certifications may no longer be current. Please see the *EC Declaration of Conformity* certificate shipped with your product.

EMC Compliance

EC Declaration of Conformity- EMC

The instrument meets intent of EC Directive 2004/108/EC for Electromagnetic Compatibility. Compliance was demonstrated to the following specifications listed in the Official Journal of the European Communities:

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

ELECTROMAGNETIC EMISSIONS:

EN 55011:2010, Radiated and Conducted Emissions Group 1, Class A [2 3]

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

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

ELECTROMAGNETIC IMMUNITY:

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

EN 61000-4-3/A2:2010 RF Radiated Electromagnetic Field, 3 V/m, 80-1000 MHz; 3 V/m, 1400 MHz - 2 GHz; 1 V/m, 2 GHz - 2.7 GHz

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

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

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

EN 61000-4-11:2004 Mains Dips and Interruptions, 0%/1 cycle, 70%/25 cycles, 0%/250 cycles [4 5]

1. To ensure compliance with all applicable EMC standards, 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 2006/95/EC for Product Safety. Compliance was demonstrated to the following specifications as listed in the Official Journal of the European Communities:

EN 61010-1:2010 Safety requirements for electrical equipment for measurement, control, and laboratory use – Part 1: General requirements

EN 61010-2:030:2010 Safety requirements for electrical equipment for measurement, control, and laboratory use – Part 2-030: Particular requirements for testing and measuring circuits

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

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

Restriction of Hazardous Substances (RoHS)

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

ISO Certification

Manufactured under an ISO 9000 Registered Quality Management System.

Warranty

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

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

The 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.

Windows License Agreement

The X-Stream 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

926367-00
September, 2015

700 Chestnut Ridge Road
Chestnut Ridge, NY 10977
USA

teledynelecroy.com