TELEDYNE LECROY
Everywhereyoulook™

Operator's Manual

HDO6000

High Definition Oscilloscopes

HDO6000 High Definition Oscilloscope Operator's Manual

922499 Rev B
November 2013

Contents

Welcome

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

The manual is arranged in the following manner:

Safety contains important precautions and information relating to power and cooling.

The sections from **Start Up** through **Maintenance** cover everything you need to know about the operation and care of the oscilloscope.

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

Remember...

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

Thank You

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

Sincerely,

David C. Graef

Teledyne LeCroy
Vice President and Chief Technology Officer

Safety Instructions

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

Symbols

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

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

High voltage. Risk of electric shock or burn.

Measurement ground connection.

Safety (protective) ground connection.

Alternating Current.

Standby Power (front of instrument).

Precautions

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

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

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

Observe all terminal ratings. Do not apply a voltage to any input (C1, C2, C3, C4 or EXT) that exceeds the maximum rating of that input. Refer to the front of the oscilloscope for maximum input ratings.

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

Use indoors only.

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

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

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

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

Operating Environment

Temperature: 5 to 40° C.

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

Altitude: Up to 3,048 m (10,000 ft) at or below 30° C.

Cooling

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

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

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

Power

AC Power

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

Power Consumption

Maximum power consumption with all accessories installed (e.g., active probes, USB peripherals, digital leadset) is 320 W (320 VA). Power consumption in standby mode is 4 W.

Ground

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

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

Start Up

Setting Up the Oscilloscope

Checking Shipment

Verify that all items on the materials list below have been shipped to you:

- 1 oscilloscope
- 4 passive probes (one for each channel)
- 1 AC line (power) cord rated for country of use
- 1 protective front cover
- 1 Getting Started Guide
- 1 Oscilloscope Security Certificate
- 1 Oscilloscope Registration Card
- 1 Calibration Document

Mixed-signal (-MS) model oscilloscopes also ship with:

- 1 digital leadset
- 5 flying ground leads
- 20 ground extenders
- 22 XL microgrippers

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.

Carrying and Placing the Oscilloscope

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

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

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

Positioning the Feet

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

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

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

Connecting to Other Devices/Systems

Make the desired cable connections. All except for the power connection are optional.

After start up, configure the connection on the oscilloscope using the menu options listed below. More detailed instructions are provided later in this manual.

Power

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

LAN

Connect a cable from either Ethernet port on the side panel to a network access device. On the oscilloscope, use the standard Windows Network dialog to configure the network connection. Go to Utilities > Preference Setup > Email to configure email settings.

USB Peripherals

Connect the device to a USB port on the front or side of the instrument. Go to Utilities > Utilities Setup > Hardcopy to configure printer settings.

External Monitor

Connect the monitor cable to a video output on the side of the instrument (VGA, DVI, and HDMI are all supported). Go to Display > Display Setup > Open Monitor Control Panel to configure the display settings.

External Controller

Connect a USB-A/B cable from the USBTMC port on the back of the instrument to the controller. Go to Utilities > Preference Setup > Remote to configure remote control.

Other Instrument (for Reference Clock)

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

Powering On/Off

The **Standby Power button** at the lower, left front of the oscilloscope controls the operational state of the instrument.

Press the button to switch the instrument into Standby mode (reduced power); press it again to return to full operation.

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

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

Always use the **File > Shutdown** menu option to execute a proper shut down process and preserve settings before powering down. Pressing and holding the Standby button will execute a "hard" shutdown, the same as on a computer, but we do not recommend doing this because it does not allow the Windows operating system to shut down properly. Do not power off by pulling the power cord from the socket or shutting off a connected power strip without first shutting down properly.

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

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

Software Activation

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

Firmware

Free firmware updates are available periodically from the Teledyne LeCroy website at teledynelecroy.com/support/softwaredownload. Registered users can receive an email notification when a new update is released. Follow the instructions on the website to download and install the software.

Purchased Options

If you decide to purchase an option, you will receive a license key via email that activates the optional features on the oscilloscope. See Add a New Software Option.

Inputs/Outputs

Front Input/Output Panel

A. The **Power button** turns on/off the oscilloscope.

B. **BNC connectors** for analog input on Channels 1–4 (or 1–2 depending on model), and **EXT** for connecting an external trigger device.

C. **Mixed signal interface** for digital inputs.

D. **Ground and calibration output terminals** are used to compensate passive probes.

E. Two (2) **front-mounted host USB ports** can be used for transferring data or connecting peripherals such as a mouse or keyboard.

Side Input/Output Panel

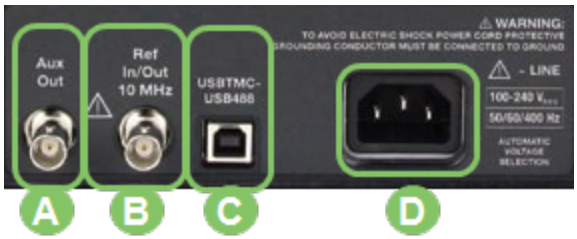
A. Video Output **VGA**, **DVI**, and **HDMI** ports connect the oscilloscope to external monitors.

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

C. **Ethernet Ports** (2) connect the oscilloscope to networks.

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

Back Input/Output Panel



A. **Aux Out** connector sends device trigger enabled, trigger out, or pass/fail output to another device.

B. **Ref In/Out** connector allows you to input an external Reference Clock, or to output a Reference Clock to another instrument.

C. **USBTMC Port** enables remote control of the oscilloscope.

D. **AC Power Inlet** connects the AC line cord.

See the general set up instructions for more information about configuring connections to other devices.

Analog Inputs

A series of BNC connectors arranged on the front and back of the oscilloscope are used to input analog signal on Channels 1-4 and AUX In, or an external trigger pulse on EXT.

HDO connectors use the ProBus interface and are compatible with any Teledyne LeCroy ProBus type probes rated for the oscilloscope's bandwidth.

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.

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

Probes

HDO6000 oscilloscopes are compatible with the included passive probes and all Teledyne LeCroy ProBus active probes that are rated for the oscilloscope's bandwidth. Probe specifications and documentation are available at teledynelecroy.com/hdo6000.

The passive probes supplied with your oscilloscope are matched to the input impedance of the instrument, but may need further compensation; refer to the probe manual for the procedure. If using other passive probes with your oscilloscope, be sure to perform a low frequency calibration using the Cal signal available from the HDO's front panel before using them to measure signal.Follow the directions in the probe instruction manual to compensate the low and/or high frequency response of the probes.

Digital Inputs

Standard with all HDO6000-MS model oscilloscopes, the digital leadset enables input of up-to-16 lines of digital data. Lines can be organized into four logical groups and can be named appropriately.

The digital leadset features two digital banks with separate threshold and hysteresis controls, making it possible to simultaneously view data from different logic families.

Connecting/Disconnecting the Leadset

To connect the leadset to the oscilloscope, push the connector into the mixed signal interface below the front panel until you hear a click.

To remove the leadset, press in and hold the buttons on each side of the connector, then pull out to release it.

Grounding Leads

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

In order to achieve optimal signal integrity, you should connect the ground at the tip of the flying lead for *each* input used in your measurements. Use either the provided ground extenders or ground flying leads to make the ground connection.

Touch Screen

The touch screen is the principal viewing and control center of the oscilloscope. The entire display area is active: use your finger or the stylus to touch, double-touch, touch-and-drag, 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 screen contains a complete menu of oscilloscope functions. Making a selection here changes the dialogs displayed at the bottom of the screen.

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

If an action can be “undone” (such as a zoom/rescale of a trace), a small 

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

Signal Display Grid

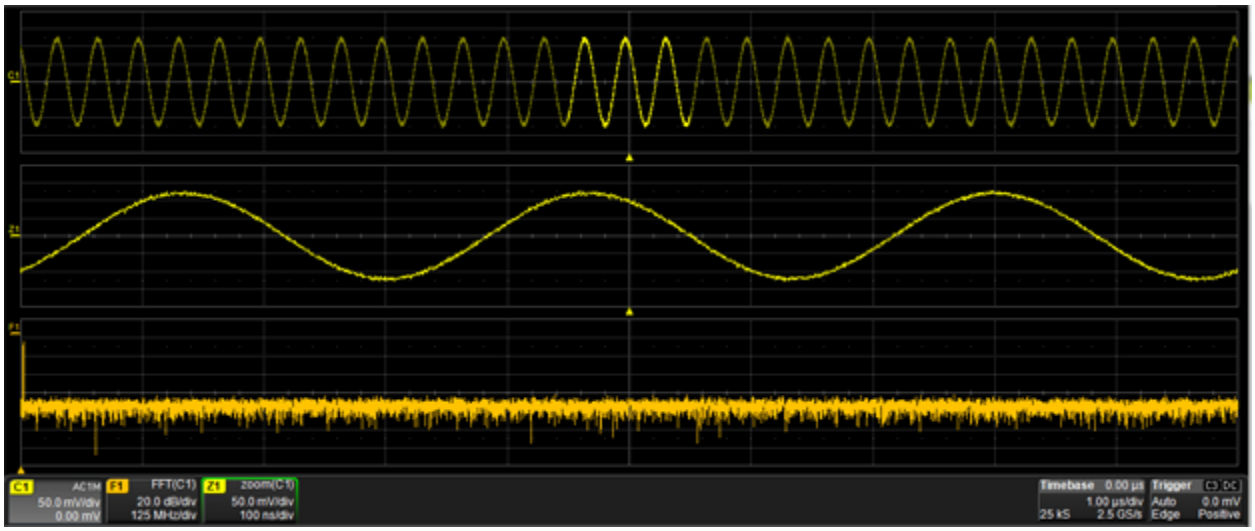
The grid area displays the waveform traces. It is sectioned into 10 Horizontal (Time) divisions and 8 Vertical (Voltage) divisions.

Multiple Grid Display

You can divide the display to simultaneously view multiple traces in different grids. By default, the oscilloscope has **Auto Grid** enabled. This divides the display into additional grids each time a new trace is opened, up to 16 grids for simultaneous viewing.

There are Display menu options to show all traces on a Single Grid, or to manually divide the display into different grid sizes and formats. When you manually divide the display, zooms and measurement markers appear on the same grid as the source channel, while math and memory traces appear in new grids until none are available.

Manually move traces from grid to grid by activating the trace and touching the **Next Grid** shortcut button. Of special note, you can also move a trace to another grid by dragging its descriptor box to the desired grid.



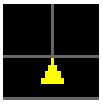
Different types of traces opening in separate grids.

Adjusting Grid Brightness

You can adjust the brightness of the grid lines to make either the grid or traces more visible. 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

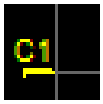
These indicators appear over the grid to mark important points on the display. They are matched to the color of the trace to which they apply.



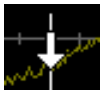
Trigger Position - A small triangle along the bottom (horizontal) edge of the grid shows the time the oscilloscope is set to trigger an acquisition. Unless Delay is set, this indicator is at the zero (center) point of the grid. Trigger Delay is shown at the top right of the Timebase descriptor box.

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

Trigger Level - This small triangle 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 - This indicator 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.

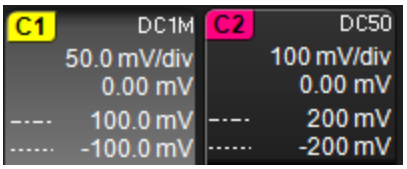
Signal Display Grid Pop-Up Menu

Touching/clicking a trace opens a pop-up menu with shortcuts to the appropriate trace setup dialog, or the Math and Measure setup dialogs. You can also use it to turn off the trace or place an annotation label on it.

Descriptor Boxes

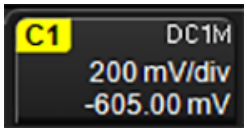
Shown just beneath the grid display, these boxes provide a summary of your channel, timebase and trigger settings. They also act as convenient navigation tools.

Descriptor boxes appear when a trace is turned on. Touch the descriptor box once to activate the corresponding trace. Touch the descriptor box a second time to open its corresponding setup dialog. When a trace is active, its corresponding descriptor Box is shown highlighted, and Front Panel controls will work for that trace.



Highlighted descriptor box (left) is active. Controls will work for this trace.

Channel Descriptor Box



Channel trace descriptor boxes correspond to analog signal inputs. They show Vertical settings and any cursor selection: (clockwise from top left) Trace Number (Cx), Pre-Processing List (summarizes changes from default state), Coupling, Gain Setting, Offset Setting, and Averaging Sweeps Count.

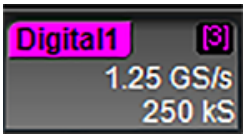
Codes are used to indicate pre-processing that has been applied to the input. The codes have a long and short form. When several processes are in effect, the short form is used.

Preprocessing Symbols on Descriptor Boxes

Pre-Processing Type	Long Form	Short Form
Sin X Interpolation	SINX	S
Averaging	AVG	A
Inversion	INV	I
Deskew	DSQ	DQ
Coupling	DC50, DC1M or AC1M	D50, D1M, or A1
Ground	GND	G
Bandwidth Limiting	BWL	B

Similar descriptor boxes appear for zoom (Zx), math (Fx), and memory (Mx) traces. These descriptor boxes show any Horizontal scaling that differs from the signal Timebase.

Digital Descriptor Box



Digital descriptor boxes appear whenever a digital line group is enabled on a mixed-signal model oscilloscope. Like Channel descriptors, they are numbered 1-4 corresponding to one of the four line groups. They show the number of digital lines in the group, the digital sample rate, and the digital memory.

Timebase 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

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

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

Dialogs

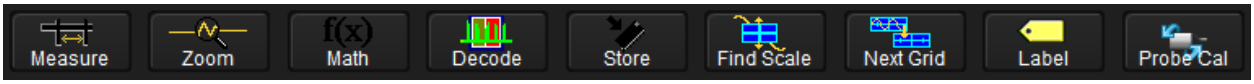
Dialogs appear at the bottom of the display for entering setup data. The top dialog will be the main entry point for the selected setup option. For convenience, related dialogs appear as a series of tabs behind the main dialog. Touch the tab to open the dialog.

Dialogs may also display right-hand dialogs (sub-tabs) or pop-up dialogs. These often change depending on the other selections made on the left-hand dialog.

Many dialog settings can be made using either the touch screen or the Front Panel buttons.

Shortcut Toolbar

Several setup dialogs contain a row of buttons at the bottom of the dialog. These provide a shortcut to common functions without having to leave the underlying set up dialog.



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

Zoom - Creates a zoom trace of the active trace.

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

Decode - Opens the main Serial Decode dialog where serial data decoders can be configured and applied.

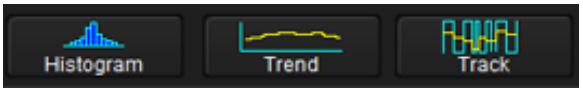
Store - Loads the active trace into the corresponding memory location (C1, F1 and Z1 to M1; C2, F2 and Z2 to M2, etc.).

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

Next Grid - Automatically 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.

The following buttons appearing at the bottom of the **Measure** (P*x*) dialogs. They allow you to create a Math function to draw the corresponding type of plot (Histogram, Trend, or Track) while remaining on the Measure setup dialog.



Control Application Window

The oscilloscope applications runs on a Windows Embedded Standard 7P Operating System and functions exactly as do other Windows applications. The application software loads automatically when you turn on the oscilloscope using the Power button.

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 oscilloscope display icon in the lower right corner of the desktop.

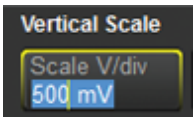
To exit the application window, choose **File > Exit**. When you exit the application, the oscilloscope operating system continues to run. To reload the application after exiting, touch the **Start DSO** desktop shortcut.

To restart Windows (reboot the oscilloscope), choose **File > Shutdown**. Wait 10 seconds then press the **Power button** on the front of the oscilloscope.

Enter/Select 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.

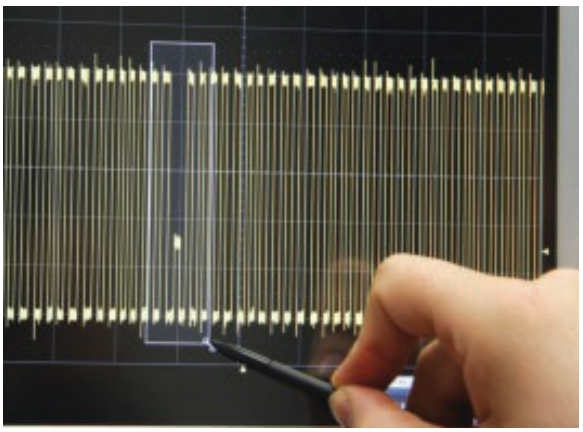


In other cases, data entry fields appear highlighted on the display. When a data entry field is highlighted, it is active and can be modified by using the Front Panel Adjust knob. If you have a keyboard installed, you can type your entry in the active field. Or, you can touch again, then select your entry from the pop-up menu or keypad.

You'll see a pop-up keypad when you double-touch 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.



Quickly zoom areas of the grid by touching and dragging to draw a selection box around a portion of the trace.

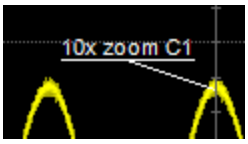
Touch & Swipe

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.

Stylus

Use the stylus when you want a more precise selection tool than your finger. It is especially helpful for selecting exact areas of the grid or values that lie close together on pop-up menus.

Annotate Traces



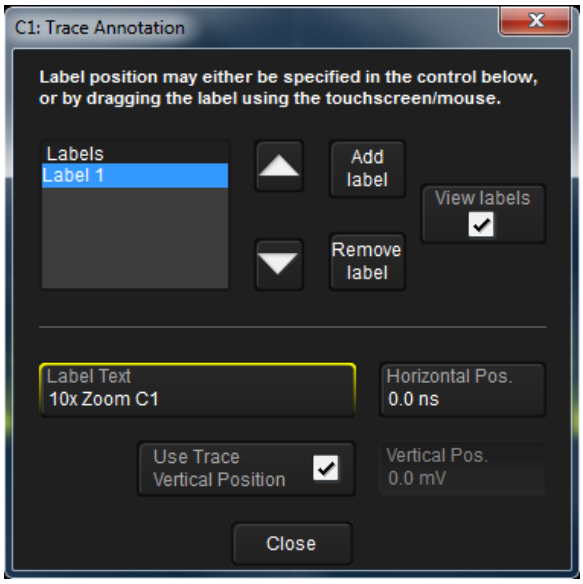
The Label function gives you the ability to add custom annotations to traces that are shown on the display. Labels are numbered sequentially in the order they were created. Once placed, labels can be moved to new positions and turned on/off.

Create Label

1. Touch the trace, then choose **Set label...** from the pop-up menu.

   OR

   Touch the trace descriptor box twice, then touch the **Label shortcut button** on the setup dialog.



2. On the Trace Annotation pop-up, touch **Add Label**.
3. Enter the **Label Text**.
4. Optionally, enter the **Horizontal Pos.** and **Vertical Pos.** at which to place the label. The default position is 0 ns horizontal. You can optionally check **Use Trace Vertical Position** instead of entering a Vertical Pos.
5. **Close** the dialog.

Edit/Remove Label

1. Touch the label and choose **Set Label** from the pop-up menu.
2. Select the **Label number**. You can use the **Up/Down arrow** keys to scroll the list.
3. Change the **Label Text** and/or **Horizontal Pos.**, or touch **Remove Label** to delete it.
4. Close the dialog.

Turn On/Off Labels

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

Print Screen

Print captures an image of the display and outputs it according to your Hardcopy settings, which may be to send it to a network printer, e-mail it, save it to a file, or copy it to the clip-board to paste into another application.

There are three ways to print:

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

NOTE: The Front Panel Print button can be configured to capture the screen as a LabNotebook entry. In this case, only the File and Utilities menu print options will function according to your Hardcopy setup.

Screen Saver

The screen saver is activated the same as on any Windows PC. Minimize the instrument display by choosing **File → Minimize** from the menu bar. Then, open the Windows Control Panel and change Appearance and Personalization settings.

Touch the oscilloscope icon at the bottom right of the desktop to restore the instrument display.

Front Panel

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

Shortcut buttons arranged across the top of the Front Panel give quick access to commonly used functions. Other shortcut buttons arranged across the bottom open special applications.

All the knobs on the Front Panel function one way if turned and another if pushed like a button. The top label describes the knob's principal “turn” action, and the bottom label describes its “push” action.

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

Front Panel Trigger Controls

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

READY and TRIG'D Indicators - The 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.

Setup - CCorresponds to the menu selection Trigger → Trigger Setup. Press it once to open the Trigger Setup dialog and again to close the dialog.

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

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

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

Stop - Prevents the 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.

Front Panel Horizontal Controls

The Horizontal Front Panel group corresponds to the Timebase dialog.

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

Horizontal Adjust knob - If the trace source is an input channel, turn this 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, turn the knob to change the horizontal scale of the trace, effectively "zooming" in or out. The value is shown on the corresponding descriptor box. Push the knob to change the setting in fine increments; push it again to return to 1, 2, 5, 10 step increments.

Front Panel Vertical Controls

Channel buttons- Turn on a channel that is off, or activate a channel that is already on. When the channel is active, pushing its channel button turns it off. A lit button shows the active channel (here, C2 is lit).

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 it once to adjust V/div in fine increments; push it again to adjust in 1, 2, 5, 10-step increments.

Dig button - Enables digital input on -MS models.

Front Panel Math, Zoom, and Mem(ory) Buttons

The **Zoom** button creates a quick zoom for each open channel trace. The resulting zoom trace (s) will be 1/10 of the channel timebase and centered on the display. Touch the zoom trace descriptor box to display the zoom controls.

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

If a Zoom, Math or Memory trace is active, the button illuminates to indicate that the Vertical and Horizontal knobs will now control that trace.

Front Panel Cursor Controls

Cursors identify specific voltage and time values on the waveform. The white cursor lines help make these points more visible, as well as provide a simple way to reposition them. A readout of the values appears on the trace descriptor box.

There are five preset cursor types, each with a unique appearance on the display: Horizontal (Time), Horizontal + Vertical, Vertical (Amplitude), Horizontal (Frequency), and Horizontal (Event). These are described in more detail in the Cursors section.

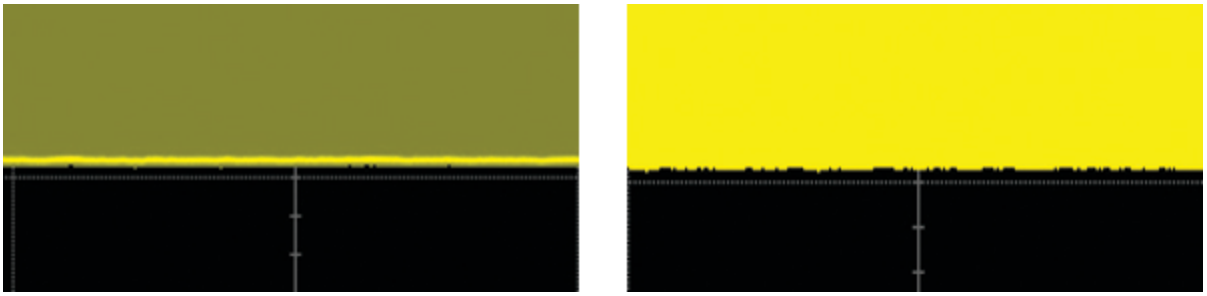
Type - Press to apply or remove cursors. Continue pressing to cycle through all cursor types until the desired type is found ("no cursors" will appear in the cycle).

Cursor knob - Turn to reposition the selected cursor line. Push to select a different cursor line to adjust.

Front Panel Adjust and Intensity Controls

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

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



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 at the same intensity.

Miscellaneous Front Panel Controls

Top Row

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

Default Setup - Resets the oscilloscope to the factory default configuration.

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

Touch Screen - Enables or disables touch screen functionality.

Clear Sweeps - Resets the acquisition counter and any cumulative measurements.

Bottom Row

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

WaveScan - Opens the WaveScan dialog.

Spectrum - Opens the Spectrum Analyzer dialog if you have that option installed.

History - Opens the History Mode dialog.

Turn On/Off Traces

Analog Traces

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

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

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

Digital Traces

From the display, choose **Vertical > Digital <#> Setup**.

From the Front Panel, press the **Dig button**, then check **Group** on the Digital<#> trace dialog. Clear Group to turn off the trace.

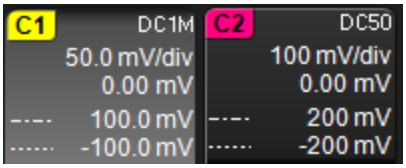
Other Traces

You can quickly create zoom or math traces without leaving the setup dialogs by touching the **Zoom** or **Math** shortcut button at the bottom of the dialog.

You can also use the Front Panel **Zoom**, **Math**, or **Mem**(ory) buttons to quickly create traces. The Zoom control automatically creates zoom trace(s) that are 1/10 of the original waveform(s). The middle of the grid is used as the center of the zoom trace.

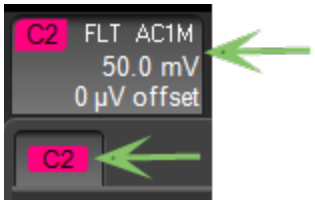
Activate Trace

A trace descriptor box appears on the display for each enabled channel, digital, zoom, math, or memory trace. Touch this box at any time to activate the trace and open its setup dialog. A highlighted descriptor box indicates the active trace to which all actions apply.



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

Although several traces may be open and appear on the grid, only one at a time is active. Whenever you activate a trace , the dialog at the bottom of the screen automatically switches to the appropriate setup dialog for that trace. The tab at the top of the dialog shows to which trace it applies.



Channel descriptor label matches Channel setup dialog tab.

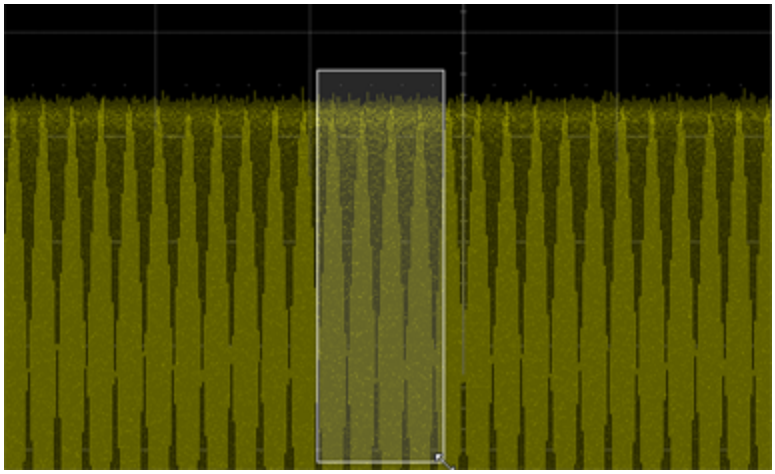
Zooming Waveforms

The Zoom function magnifies a selected region of a trace. On HDO6000 model oscilloscopes, you can display up to eight zoom traces (Z1 - Z8) taken from any channel, math, or memory trace.

You can also use the Multi-Zoom Math function to create time-locked zoom traces for selected waveforms. For more information, refer to Multi-Zoom).

Create Zoom

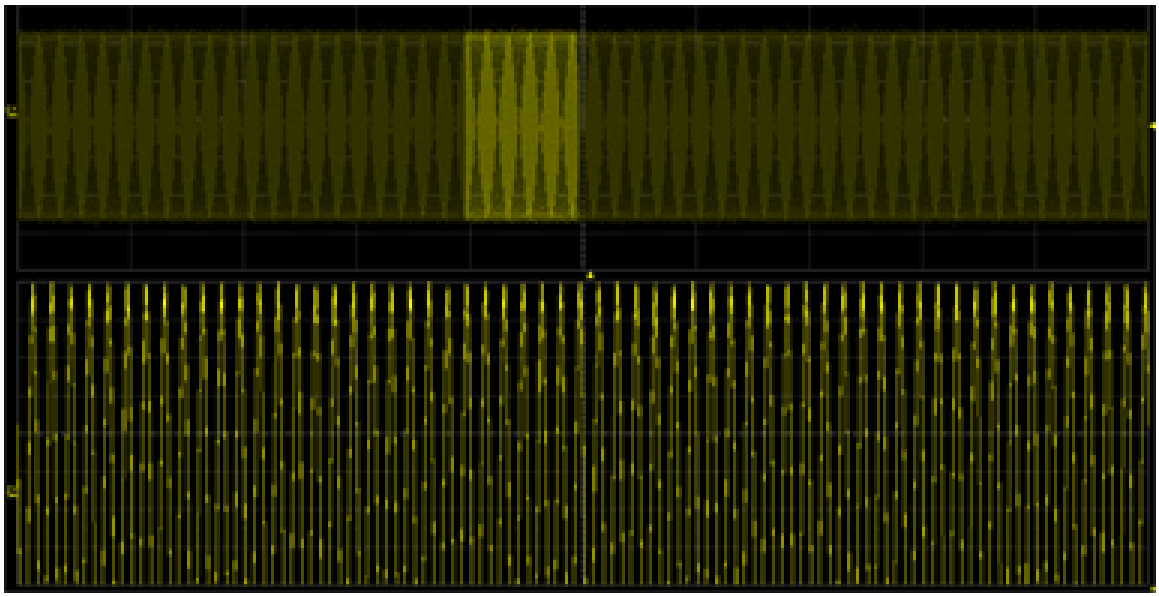
To create a zoom, touch -and-drag to draw a selection box around any part of the source waveform.



Selected portion of trace.

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

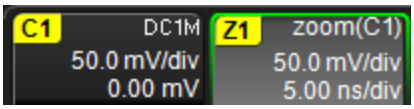
The zoom opens in a new grid, or the next empty grid, with the zoomed portion of the source trace highlighted. If there are no more available grids, zooms will open in the same grid as the source trace.



Zoomed area of original trace highlighted.

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 location, ready to be turned on again when desired.

The zoom's Vertical and Horizontal units will differ from the source trace, as seen from a comparison of the trace descriptor boxes, because the zoom is showing a scale, not a measured level.



Channel descriptor box and its Zoom descriptor box.

You can further adjust these settings using the Front Panel knobs, or by changing the settings on the **Zoom dialog**. Touch the zoom descriptor box to activate it, then touch it again to display the **Z*x* tab**.

Because it is a calculated and not a sampled trace, you can adjust the zoom's Horizontal Scale without changing the oscilloscope's Timebase (a characteristic shared with math and memory traces).

Turn off Zoom

Turn off a zoom trace the same as you would any other trace:

- Deselect the **Trace On** checkbox on the Zoom dialog.
- Touch-and-hold (right-click) the descriptor box until the pop-up menu opens, then choose **Off**.

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.

Multi-Zoom

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.

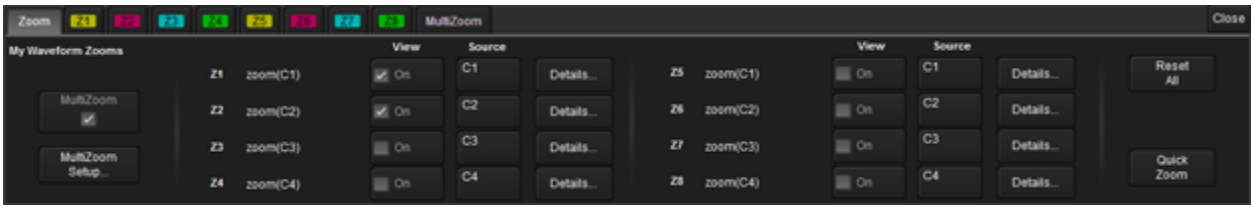
Zoom Controls

Once the zoom trace has been created, adjust its Vertical and Horizontal Scale to further "zoom" in or out. You can do this by activating the zoom trace and using the Front Panel Vertical and Horizontal knobs, or by modifying settings on the **Zoom dialog**.

To access it the Zoom dialog, double-touch any zoom trace 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** - returns all zooms to x1 magnification.
- **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-Z8). Each dialog reflects the current scale settings for that zoom.

Trace Controls

Trace On - displays the zoom trace. Select/deselect this box to show/hide the zoom.

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 only in Sequence Sampling Mode.

Rescale Controls

These controls on the **Zx dialogs** are the same used to rescale any trace, and you will see them throughout the oscilloscope software. They work the same wherever they appear.

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

Var.checkbox - enables variable zooming in increments finer than the default 1, 2, 5, 10 step increments. When checked, each touch of the zoom control buttons changes the degree of magnification by a single increment.

Horizontal Scale/div- sets the amount of time represented by each horizontal division of the grid. It is the equivalent of Time/div, only unlike the Timebase setting, it may be set differently for each zoom, math function, or memory trace.

Vertical Scale/div - sets the voltage level represented by each vertical division of the grid; it's the equivalent of V/div used for channel settings.

Horizontal/Vertical Center - sets the voltage or time that is to be at the center of the screen on the zoom trace. The horizontal center is the same for all zoom traces.

Reset Zoom - returns the zoom to x1 magnification.

Rescale Memory or Math Function Traces

Unlike channel traces, memory (M1 - M4) or math function (F1 - F8) traces can be rescaled directly without having to create a separate zoom trace. The same set of controls used to rescale 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 those that are generated through analysis options.

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 in which to draw the trace, Z1-Z8, but the source trace remains at the original scale.

Multi-Zoom

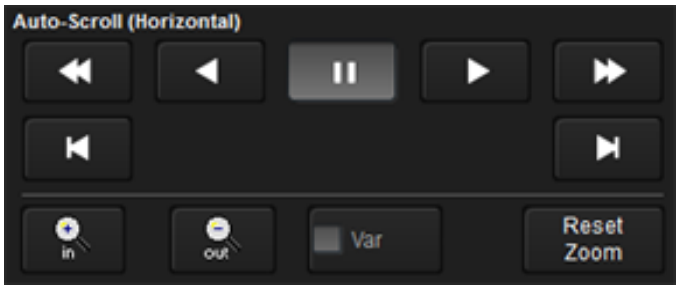
Multi-Zoom creates time-locked zoom traces for only the waveforms that you choose to include. The zooms are of the same X-axis section of each waveform. As you scroll through a waveform, all included zooms scroll in unison.

Set Up Multi-Zoom

1. Choose **Math → Zoom Seutp...** to open the Zoom dialog, then touch the **Multi-Zoom tab** or **Multi-Zoom Setup...** button.
2. On the Multi-Zoom dialog, turn **Multi-Zoom On** and 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 together in time-locked steps from the beginning to the end of the acquisition.



They are (from left to right, top to bottom row):

Scroll Left Fast - back in time

Scroll Left Slow - back in time

Pause - stop scrolling

Scroll Right Slow - forward in time

Scroll Right Fast - forward in time

Jump to Start - go to beginning of acquisition

Jump to End - go to end of acquisition

In/Out - increase or decrease magnification level of zooms

Var - zoom In/Out in finer increments than the default 1, 2, 5, 10 steps

Reset Zoom- return all zooms to same scale as the source trace.

Turn Off Multi-Zoom

1. From the menu bar, touch **Math → Zoom Setup...**.
2. On the main Zoom dialog, **deselect the MultiZoom checkbox**.

Vertical

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

NOTE: While Digital settings can be accessed through the Vertical menu on -MS model oscilloscopes, they are handled quite differently. See Digital Overview.

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 (C*x*) always shows the current Vertical Scale setting.

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

The Channel dialog contains:

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

If a probe is connected to the channel, the Channel dialog also contains a tab for the Probe dialog.

Vertical Settings

Vertical Scale - Set the vertical scale or sensitivity, and choose fixed or variable gain adjustment.

Vertical Offset - Select between zero vertical offset or to set the offset to a specific value.

Coupling - Select from DC 50 Ω, DC1M, AC1M and GROUND.

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

Bandwidth - Bandwidth filters are available at a variety of fixed bandwidth settings. The exact settings vary by model.

Probe Attenuation -Enables you to set probe attenuation manually if using a third-party probe. The oscilloscope's inputs automatically sense Teledyne LeCroy probes and sets probe attenuation and Vertical units for you.

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

Averaging - 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.

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 Pre-processing deskew and the Math deskew functions perform the same activity. See Deskew Channels.

Invert - Inverts the waveform for the selected channel.

Interpolation - 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 wave shapes, especially when the sample rate is 3 to 5 times the system bandwidth.

Noise Filter (ERes) - Enhanced Resolution (ERes) filtering increases 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.

Probe Dialog

The Probe Dialog displays probe attributes and (depending on the probe type) allows you to AutoZero or DeGauss Teledyne LeCroy probes from the oscilloscope touch screen. When a probe is not connected, the Channel dialog shows only the C1 tab for vertical setup.

Channel dialog with tab for connected probe.

Probe Information on Channel Dialog

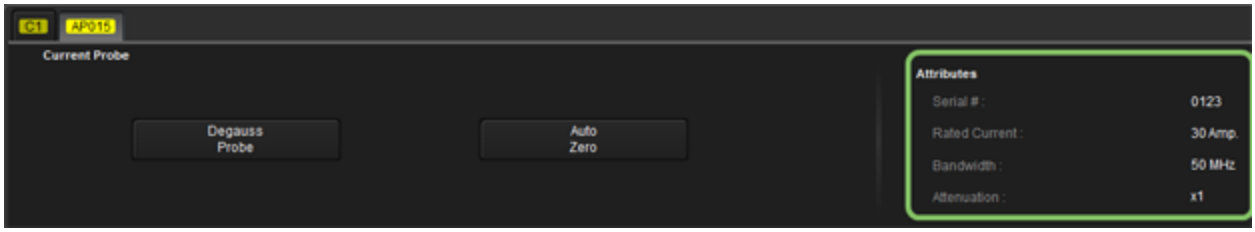
After a Teledyne LeCroy probe is connected, it is recognized by the oscilloscope.

- For passive probes, attenuation is automatically set, and these fields are disabled on the Channel setup dialog.
- For active voltage and current probes, an additional tab with the probe model name is displayed to the right of the C1 tab. Click on the tab to display the probe dialog.

When using third-party probes, the Probe Attenuation and Deskew values may be entered manually on the Channel dialog.

Probe Information on Probe Dialog

This additional tab contains specific information on the connected probe. Default values for the probes coupling and attenuation are automatically downloaded from the probe, and these settings along with other attributes are shown on the dialog. Other controls may be available depending on the probe model or input device type.



Probe dialog showing the connected probe's control attributes.

Deskew Channels

The signal input channels are deskewed at the factory prior to shipment and should not require any further action.

Follow this generic deskew procedure to compensate for propagation delays due to different lengths of cables, probes, or anything else that might cause timing mismatches between signals.

1. Connect all probes to the desired channels, then probe a common signal with each probe.
2. Turn on two channels, one of which will be the reference channel for the entire deskew procedure.
3. Set an Edge trigger on the reference channel.
4. Switch to **Display > Single Grid** so both traces appear on the same grid.
5. Turn the Front Panel Horizontal knob to adjust **Time/div** so that you can clearly see the edges of each trace.
6. Touch the **channel descriptor box** for the second channel twice to open the setup dialog.
7. Touch the **Deskew** field to activate it.
8. Turn the Front Panel **Adjust knob** until the trace aligns with the reference waveform.
9. Repeat Steps 2 through 8 for each input, using the same channel as the reference and same trigger each time.

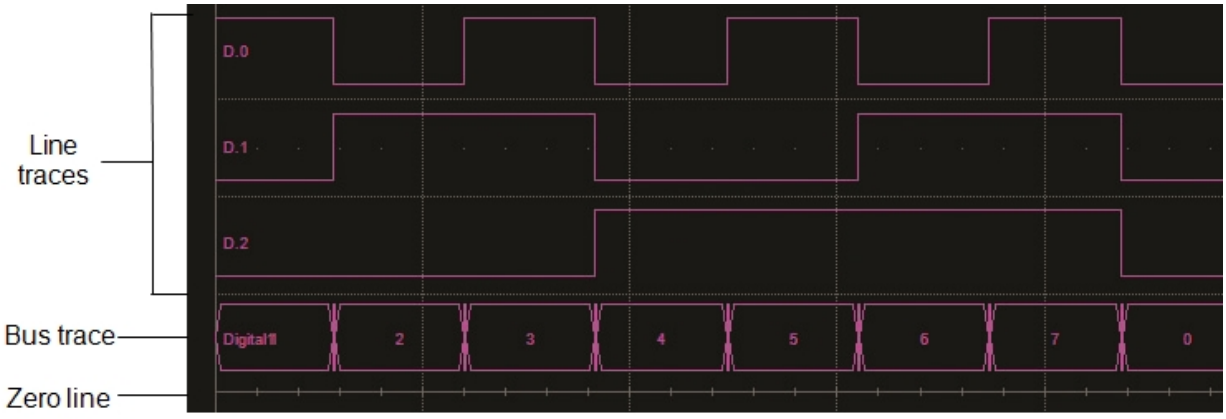
Digital (Mixed Signal)

The digital leadset (standard with -MS model oscilloscopes) inputs up-to-16 lines of digital data. Leads are organized into two banks of eight leads each, and you assign each bank a standard Logic Family or a custom Threshold and Hysteresis to capture the digital signals.

The Digital set up dialog has four tabs each corresponding to one of four possible digital groups, labeled Digital1 to Digital4. You choose which lines from among the 16 make up each digital group, what they are named, and how the group appears on the display. Initially, logical lines are numbered the same as the physical lead they represent, although any line number can be re-assigned to any lead.

Digital Traces

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

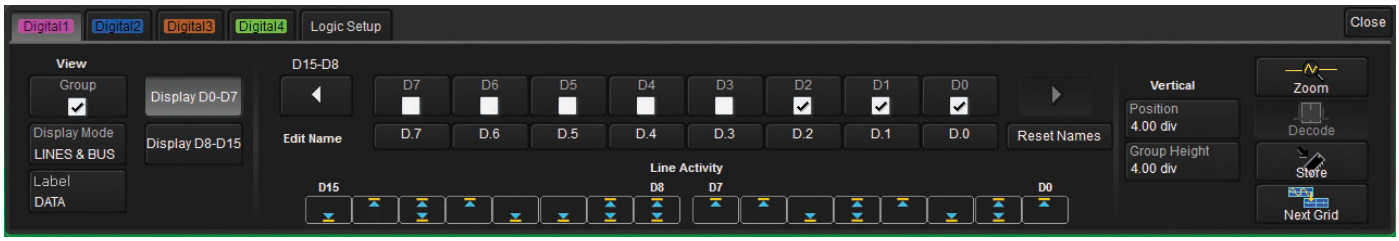


Three Digital Line traces and a Bus trace displayed with a Vertical Position of positive 4.0 divisions (top of grid) and a Group Height 4.0 divisions (half the grid).

Activity Indicators

Activity indicators at the bottom of the Digital<#> dialogs show which lines are High (up arrow), Low (down arrow), or Transitioning (up an down arrows) relative to the Logic Threshold value. They provide a quick view of which lines are active and of interest to display on screen.

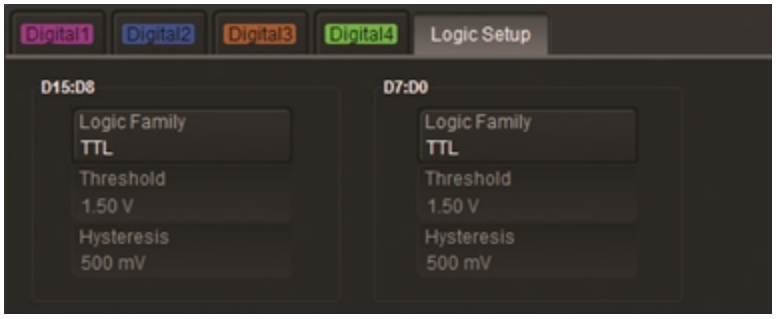
Digital Group Set Up



1. From the menu bar, choose **Vertical > Digital <#> Setup**, or press the Front Panel **Dig button** and select the desired **Digital<#> tab**.
2. On the Digital<#> set up dialog, check the boxes for lines **D0** through **D15** that comprise the group.

   Touch the Display D0-D7 and Display D8-D15 buttons to quickly turn on the entire digital bank, or touch the Right and Left Arrow buttons to switch between each digital bank as you make line selections.

   NOTE: Each group can consist of anywhere from 1 to 16 of the leads that are (or will be) connected to signal, from either digital bank regardless of the Logic set on the bank. It does not matter if the some or all of the lines have been included in other groups.
3. When all group members are selected, optionally rename them.
4. Go on to set up the digital display for the group. Check **Group** to enable the display.
5. When you're finished on the Digital<#> dialog, touch the **Logic Setup tab** and choose the **Logic Family** that applies to each digital bank, or set custom **Threshhold** and **Hysteresis** values.



Digital Display Set Up

You can choose the type and position of the digital traces that appear on screen for each digital group.

1. Set up the digital group.
2. Touch **Display Mode** and choose from:
   - **Lines** - the default display, which shows a time-correlated trace indicating high, low, and transitioning points (relative to the Threshold) for *every* digital line in the group. The size and placement of the lines depend on the number of lines, the Vertical Position and Group Height settings.
   - **Bus** - which collapses the lines in a group into their Hex values. It appears immediately below all the Line traces when both are selected.
   - **Lines & Bus** - which displays both line and bus traces at once.

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

   In the example above, the first Line trace (D0) starts at positive 4.0 divisions off the zero line, or at the very top of the eight vertical division grid.

4. In **Group Height**, enter the total number of grid divisions the entire display should occupy. All the selected traces (Line and Bus) will appear in this much space.

   Individual traces are resized to fit the total number of divisions available. The example above shows a group of three Line traces plus the Bus trace occupying a Group Height of 4.0 divisions. Each trace takes up one division.

5. Check the **Group** box to enable the display.

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

   To close traces, uncheck the **Group** box , or touch-and-hold on the **Digital<#> descriptor** box and choose **Off** from the pop-up menu.

Renaming Digital Lines

The labels used to name each line can be changed to make the user interface more intuitive. Also, labels can be "swapped" between lines.

Changing Labels

1. Set up the digital group.
2. Touch **Label** and select from:
   - **Data** - the default, which appends "D." to the front of each line number.
   - **Address** - appends "A." to the front of each line number.
   - **Custom** - lets you create your own labels line by line.
3. If using Custom labels:
   - Touch the **Line number button** below the corresponding checkbox. If necessary, use the **Left/Right Arrow buttons** to switch between banks.
   - Use the virtual keyboard to enter the name, then press **OK**.

   The button and any active line traces are renamed accordingly.

Swapping Lines

This procedure helps in cases where the physical lead number is different from the logical line number you would like to assign to that input (e.g., a group is set up for lines 0-4, but lead 5 was accidentally attached to the probing point). It can save time having to re-attach leads or re-configure groups.

1. Select a **Label** of **Data** or **Address**.
2. Touch the **Line number button** below the corresponding checkbox. If necessary, use the **Left/Right Arrow buttons** to switch between banks.
3. From the popup, choose the line with which you want to swap labels.

   The button and any active line traces are renumbered accordingly.

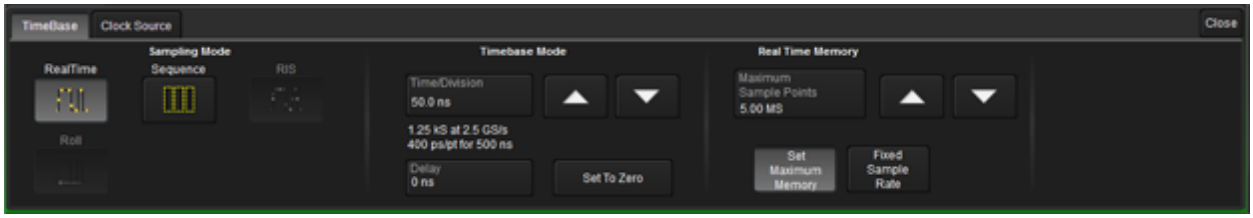
Timebase

Timebase, also known as Horizontal, settings control the trace along the X axis. These settings are shared by all channel traces.

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 main Timebase dialog contains settings for Sampling Mode, Timebase Mode, and Real Time Memory. Related tabs open dialogs to set up Sequence Mode and Clock Source.

Timebase Settings



Sampling Mode

Real Time, Sequence, Roll, or RIS mode.

Timebase Mode

These controls set the timebase shared by all channels.

Time/Division - 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 - The 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 set how the oscilloscope samples when in Real Time mode.

Sampling Rate - 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 oscilloscope's 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.

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

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

Capture Interval = 1/Sample Rate x Memory

Capture Interval/10 = Time Per Division

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

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

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

Roll Mode

Roll mode displays, in real time, incoming points in single-shot acquisitions that have a sufficiently low data rate. This mode can be invoked for slow acquisitions where the time per division is 100 ms/div or slower. Roll mode samples at ≤ 2.5 MS/s.

The oscilloscope appears to "roll" the incoming data continuously across the screen 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.

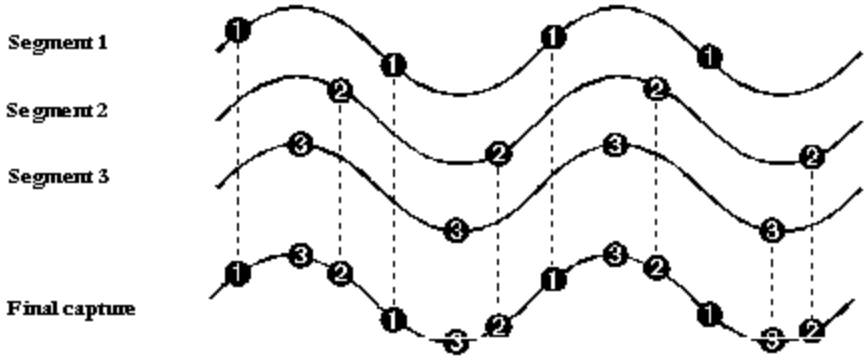
NOTE: If the processing time is greater than the acquire time, the data in memory is overwritten. In this case, the instrument issues the warning, "Channel data is not continuous in ROLL mode!!!" and rolling starts again.

RIS Sampling Mode

RIS (Random Interleaved Sampling) is an acquisition technique that 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 8

ps (125 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.

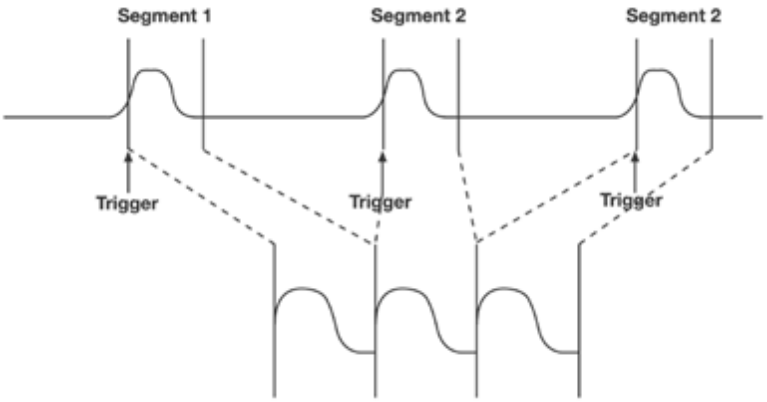


Interleaving of sample in RIS sampling mode.

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 oscilloscope uses the sequence timebase setting to determine the capture duration of each segment as 10 x time/div. With this setting, the oscilloscope uses the desired number of segments, maximum segment length, and total available memory to determine the actual number of samples or segments, and time or points.

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

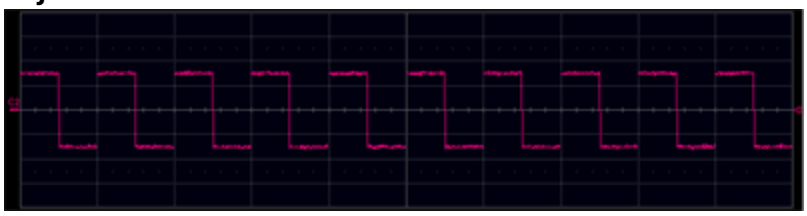


Capturing segments in Sequence sampling mode.

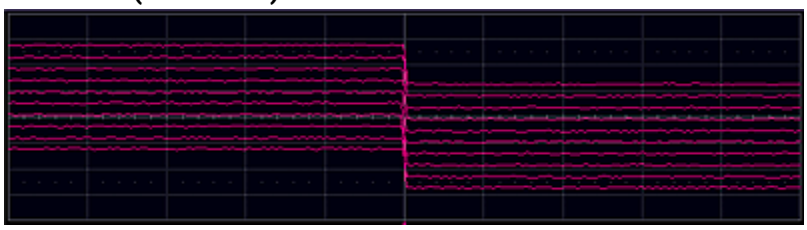
Sequence Display Modes

The instrument gives you a choice of five ways to display your segments:

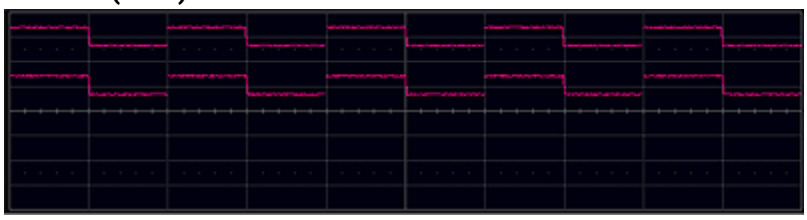
- **Adjacent**



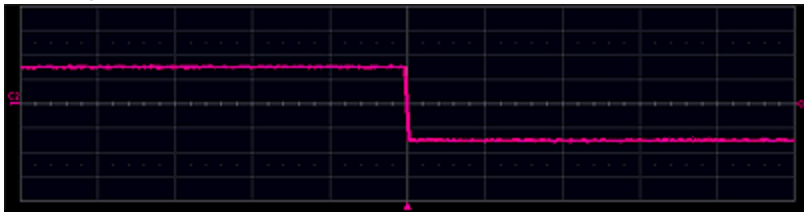
- **Waterfall (cascaded)**



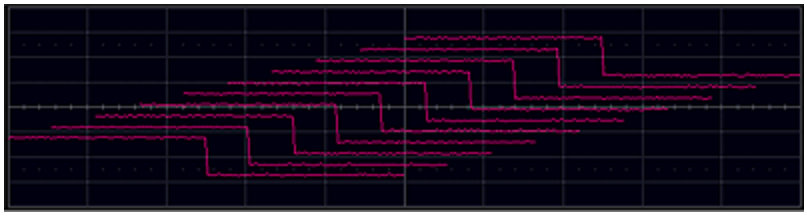
- **Mosaic (tiled)**



- **Overlay**



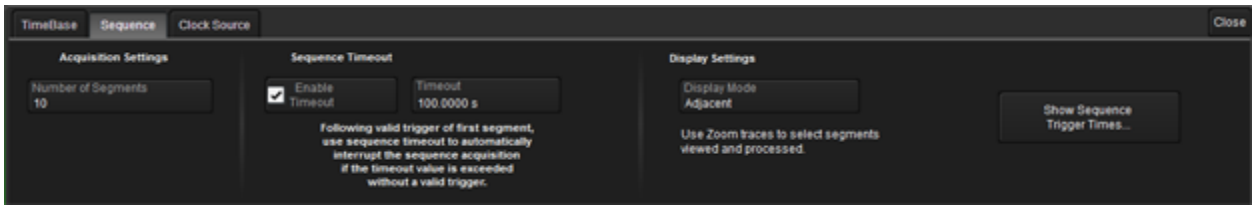
- **Perspective**



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 oscilloscope uses the sequence timebase setting to determine the capture duration of each segment. Along with this setting, the oscilloscope uses the number of segments, maximum segment length, and total available memory to determine the actual number of samples or segments, and time or points.



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

   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 will complete in a reasonable amount of time and control of the oscilloscope will return to the operator/controller without having to manually stop the acquisition.
5. Touch **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.

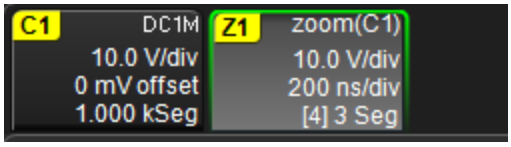
   NOTE: Once acquisition has started, you can interrupt it at any time by pressing the **Stop** Front Panel button. In this case, the segments already acquired will be retained in memory.

View Segments In Sequence Mode

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

Tip: By changing the Num field value to 1, you can use the Front Panel Adjust knob to scroll through each segment in order.

Channel descriptor boxes indicate the total number of segments acquired. Zoom descriptor boxes show the number of 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 may have acquired 1000 segments. You chose to display segments number 4-6. The Channel descriptor box will read 1000. The Zoom descriptor box will read [4]3.

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 (Fx) tab** to display its corresponding dialog.
3. On the dialog, touch **Operator1** and select the **Segment** button from the pop-up menu.
4. Touch the **Select** right-hand dialog tab.
5. Touch **First Selected** and choose the first segment to display.
6. Touch **Number of Selected** and enter the number of segments to display at once.

View Segment Time Stamps

To view time stamps for each segment:

1. From the menu bar, choose **Timebase → Acquisition Status** or **Vertical → Channel 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.

Clock Source Settings

Sample Clock

These settings determine the clock that controls when the oscilloscope's digitizers sample the input waveforms. The default setting is to use the oscilloscope's **Internal** clock. To use an external sample clock:

1. Connect the clock source to the **Ext** input on the front of the oscilloscope using a BNC cable.
2. Go to **TImebase > Horizontal Setup** and choose **Real-time Sampling Mode**.
3. On the **Clock Source tab** under **Sample Clock** choose from 0V, ECL, TTL.
4. Choose an **External Coupling** of 50 Ohms, Ground, or 1 M Ohm.

Reference Clock

These settings control the Timebase reference used to synchronize acquisition across all channels. The default setting is to use the oscilloscope's **Internal** 10 MHz clock. To use an external reference clock:

1. Connect the clock source to the **Ref In/Out 10 MHz** input on the back of the oscilloscope using a BNC cable.
2. Go to **Timebase > Horizontal Setup** and choose **Real-Time Sampling Mode**.
3. On the **Clock Source tab** under Reference Clock choose **External**.

External Reference Clock vs. External Sample Clock

An external reference clock is used to synchronize the oscilloscope's internal timebase to an external frequency source. This allows multiple instruments to lock their timebases to a common source.

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

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

Auto Setup

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

Vertical Scale (V/div), Offset, Timebase (Time/div), and Trigger are set so that there is an Edge trigger on the first, non-zero-level amplitude, and the entire waveform is visible for at least 10 cycles over the 10 horizontal divisions.

To run Auto Setup, you can either:

- Press the **Auto Setup** Front Panel button.
- Choose **Auto Setup** from the **Vertical**, **Timebase**, or **Trigger** menus. All these options perform the same Auto Setup function.

To confirm Auto Setup, press the Auto Setup button again or use the touch screen display.

Restore Default Setup

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

Default settings for your oscilloscope include the following:

Channel/Vertical	C1-C4 on at 50 mV/div Scale, 0 V Offset, Linear Interpolation
Timebase	Real Time Sampling at 50 ns/div, 0 Delay, 1.25 kS at 2.5 GS/s, 1.0 MS Memory
Trigger	C1 with an Auto Positive Edge, DC Coupling, 0 V Level
Display	Auto Grid
Cursors	Off
Measurements	Cleared
Math	Cleared

Trigger

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

All trigger types allow for pre-trigger or post-trigger delay, the display time relative to the trigger event (although the trigger itself may not be visible), or let you set the time between sweeps, how often the display is refreshed. Unless modified by a pre- or post-trigger delay, the trigger event occurs at point zero at the center of the grid, and an equal period of time before and after this point is shown to the left and right of it.

Trigger capabilities include:

- **Simple Triggers** - activated by basic waveform features such as an edge with a positive or negative slope or width.
- **Pattern Triggers** - trigger the oscilloscope when a pattern condition, from false to true, occurs on selected input channel and external input.
- **SMART Triggers** - sophisticated triggers that enable you to use basic or complex conditions for triggering. Use SMART Triggers for signals with rare features, like glitches.
- **Measurement Trigger** - triggers that allow you to leverage parameter measurements as waveform trigger conditions. A measurement trigger is either the only trigger or the final trigger in a chain of trigger events including hardware triggers.
- **MultiStage Triggers** - varied forms of triggers including **Cascaded**, **QualFirst**, and **Qualified** allowing varied combinations of triggers and trigger stages.
- **Serial Triggers** - provide triggers specific to a wide variety of serial data protocols.
- **TV Triggers** - provide the ability to trigger on multiple types of video signal.

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

Trigger Modes

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

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

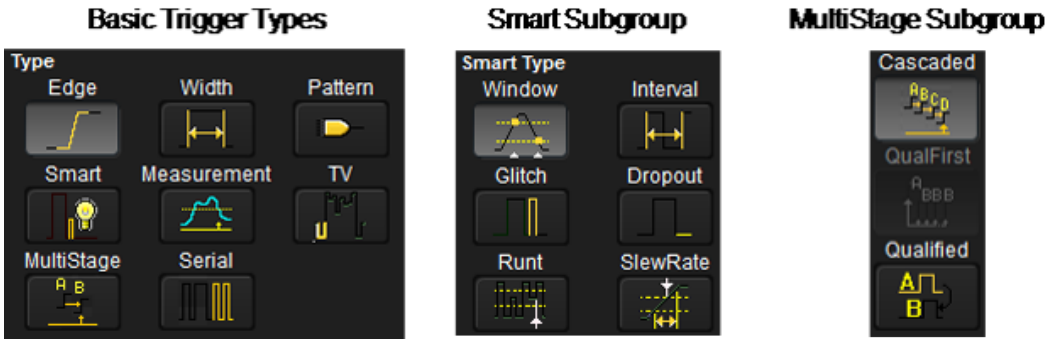
In **Normal** mode, the oscilloscope sweeps only if the input signal reaches the set trigger point. Otherwise it continues to display the last acquired waveform.

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

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

Trigger Types

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



Basic Triggers

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

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

Pattern triggers on a logical combination of analog or digital inputs: CH1, CH2, CH3, CH4, EXT, and D0-D15. You have a choice of four Boolean operators (AND, NAND, OR, NOR) and can stipulate the high or low voltage logic level for each input independently.

NOTE: Only the AND Boolean operator is available when combining analog and digital inputs.

Measurement triggers when a certain parameter measurement is found. A measurement trigger is either the only trigger or the final trigger in a chain of trigger events including hardware triggers.

TV triggers on standard (PAL, SECAM, NTSC, HDTV) or custom composite video signals.

Smart Triggers

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

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

Glitch triggers upon finding a fixed pulse-width time or time range.

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

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

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

MultiStage Triggers

A type of MultiStage trigger, **Cascaded** triggers when a succession of criteria in Stages A-D are met. Each stage can result in different trigger actions, such as arm only, trigger only, or trigger and rearm.

QualFirst arms the oscilloscope on the A event, then triggers on all subsequent B events.

NOTE: This button is enabled when using the sequence sampling mode. It is commonly used in sequence mode for disk drive applications with the index pulse defined as the A qualifier signal and the servo gate signal as the B triggering events.

Qualified arms the oscilloscope on the A event, then triggers on the B event. In Normal trigger mode, it automatically resets after the B event. A (arm) can be Edge, Pattern, State, or PatState events; B (trigger) can be Edge or Pattern events.

Only available as a sub-type of Qualified triggers, **PatState** triggers when the qualifying signal goes above or below a specified voltage level. You can specify the number of these events that must occur to trigger.

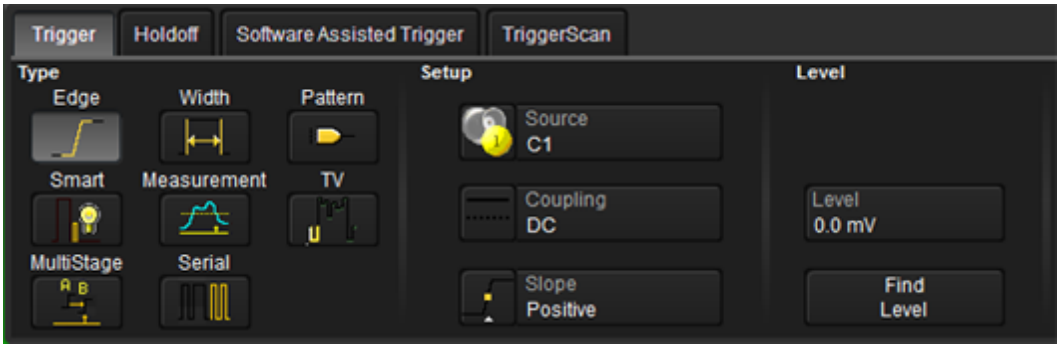
Serial Triggers

Protocol-enabled serial triggers are available as options on some oscilloscope models. This trigger type will be available only if you have such an option installed.

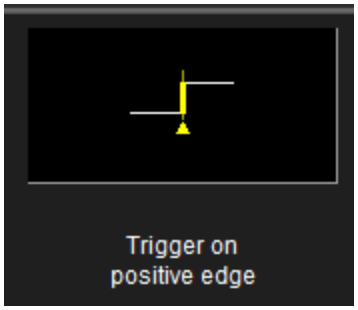
Trigger Settings

To access the Trigger setup dialogs, choose **Trigger > Trigger Setup** from the menu bar, or press the **Front Panel Trigger Setup** button.

The dialogs you see and the options on them will vary depending on your trigger type selection. The main Trigger dialog contains settings that are required for most trigger types.



The trigger setup is summarized in a preview window at the far right of the Trigger dialog.



Type - see Trigger Types for details. This selection drives the remainder of the trigger setup. The default selection is Edge.

Source - the channel signal upon which to base the trigger. If a trigger is designed to work with multiple inputs, like a Pattern trigger, you do not have to choose a single source, but will be given controls for setting the conditions on each source.

Coupling - 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).

Level - the source Voltage level or levels that mark the threshold for the trigger to fire. Trigger levels specified in Volts normally remain unchanged when the vertical gain or offset is modified.

Find Level - where available, this button sets the Level to the signal mean.

Trigger Holdoff

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

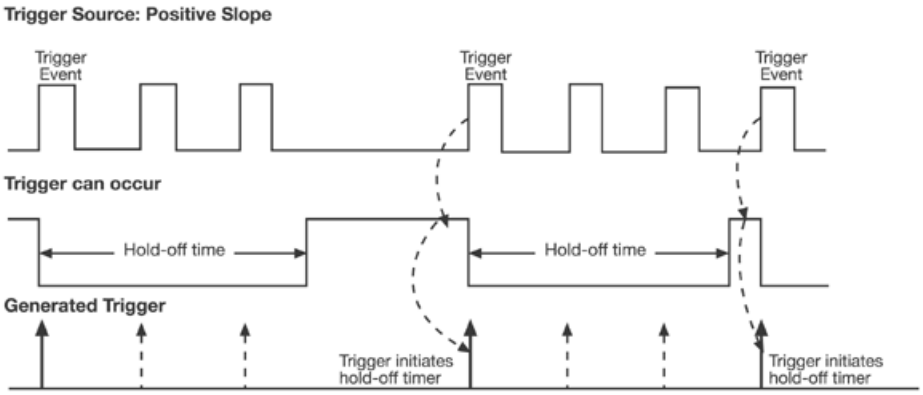
Use holdoff to obtain a stable trigger for repetitive, composite waveforms. For example, if the number or duration of sub-signals is known, you can disable them by choosing an appropriate holdoff value. Qualified triggers operate using conditions similar to holdoff.

Hold Off by Time

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

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

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

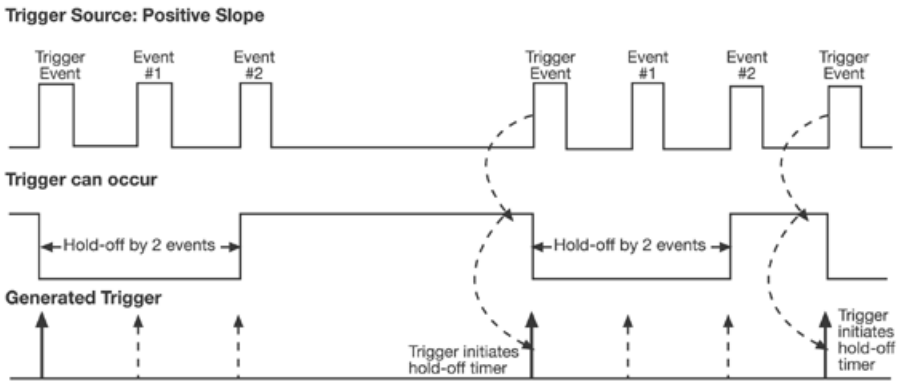


Edge trigger with holdoff by time.

Hold Off by Events

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

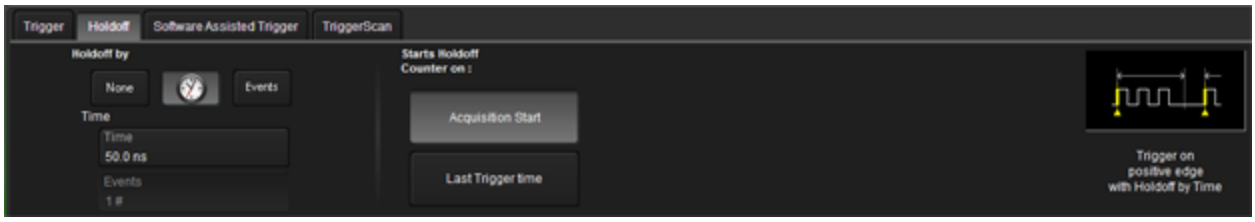
In the figure below, the bold edges on the trigger source indicate that a positive slope has been selected. The broken, upward-pointing arrows indicate potential triggers, while the bold ones show where triggers actually occur after the holdoff expires.



Edge trigger with holdoff by events.

Holdoff Settings

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



Holdoff by - type of holdoff to use with trigger: None, Time (clock), or Event.

Time - if using Holdoff by Time, the time in S to wait before triggering.

Events - if using Holdoff by Events, the number of events to count before triggering.

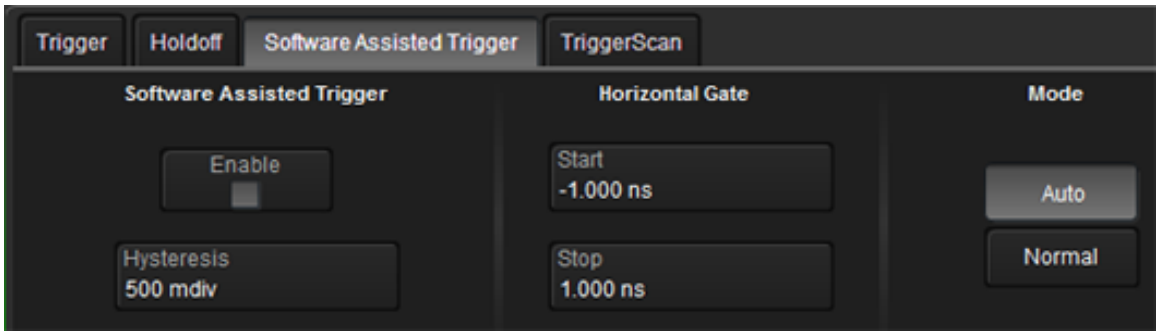
Starts Holdoff Counter On - whether to count holdoff time/events from Acquisition Start or Last Trigger Time before triggering again.

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**.

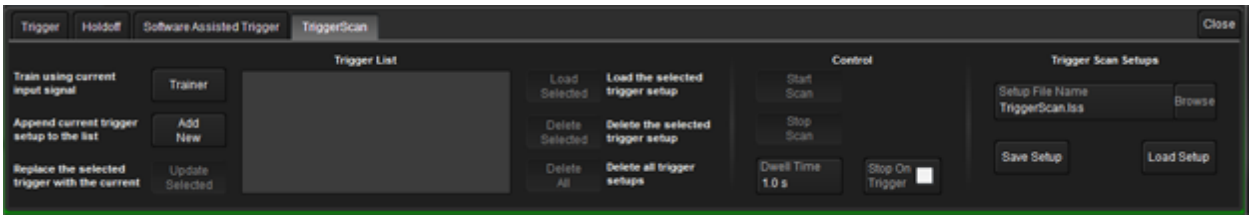


2. Touch **Enable**.
3. Create a trigger window by entering a **Hysteresis** value. This value sets a boundary above and below the main trigger level to exclude noise.
4. Choose **Auto** or**Normal**; this determine the trigger behavior when trigger crossings are not found in the trigger source waveform.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

TriggerScan

TriggerScan is a debugging tool (available for any trigger type) that helps you quickly find rare waveform glitches and anomalies. With TriggerScan, you can build a list of trigger setups to look for rare events and automatically sequence through each one. TriggerScan can use any type of trigger setup available including edge, width, and qualify as well as Smart Triggers (such as, glitch and runt triggers). TriggerScan automates two key processes in triggering rare events:

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



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 touch the **TriggerScan** tab.
2. Touch the **Trainer** button.
3. Choose the **Source** channel on which to train, and select all the trigger types you want to set up.
4. Touch the **Start Training** button. The training begins. When it is complete, a list of smart trigger setups is displayed in the Trigger List.

Modify Trigger List

The Trigger List displays a list of of the triggers created by the Trainer. Follow these steps to add or remove triggers, or update their individual setups. Once you have made any changes to the Trigger List, you are ready to start scanning.

1. If not already there, choose **Trigger → Trigger Setup...** from the menu bar, then touch the **TriggerScan** tab.
2. Make any of the following modifications to the Trigger List:
   - To add a new trigger setup to the list, touch the **Trigger** tab 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**.

- To replace a Trigger List setup with the setup on the Trigger dialog, highlight the setup in the Trigger List and touch the **Update Selected** button.
- To use a trigger from the Trigger List, highlight its corresponding row on the list, and then touch the **Load Selected** button.
- To delete a trigger setup, highlight the setup in the Trigger List and touch the **Delete Selected** button.

All trigger setups can be deleted regardless of selections on the Trigger List with one step by touching the **Delete All** button.

3. Once you have made the desired changes to the Trigger List, touch the **Trainer button** and restart the scan by touching the **Start Training** button on the Trigger Scan Trainer pop-up. The oscilloscope automatically cycles through all the trigger setups.

NOTE:

- Use **Dwell Time** to tune the time that the oscilloscope waits before loading the next trigger.
- If you want TriggerScan to stop when the oscilloscope next triggers, check the **Stop On Trigger** checkbox. You can use this to isolate trigger setups.
- If you have Persistence enabled, all trigger events are recorded on the display.

Saving TriggerScan Setups

Save TriggerScan setups whenever you have modified the Trigger List. 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.

NOTE: You can load previously saved TriggerScan setups by touching the **Browse** button, locating the file, then touching **Load Setup...**.

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.

See Utilities for settings related to screen resolution, color, and screen savers.

Display Setup

Follow this procedure to adjust how traces appear on the touch screen display.

1. From the menu bar, choose **Display → Display Setup...**
2. Touch the **Grid button**, then select one of the grid types (the image on the icon shows the resulting grid arrangement).

   Auto, the default, automatically adds or deletes grids as you open or close traces, up to the maximum number supported.
3. To dim or brighten the background grid lines, touch **Grid Intensity** and provide a value from 0 to 100.
4. Optionally, check **Grid on top** to superimpose the grid over the waveform.

   NOTE: Depending on the grid intensity, some of your waveforms may be hidden from view when the grid is placed on top. To view them, simply uncheck Grid on top.
5. Optionally, check **Axis labels** to display the values of the top and bottom grid lines (calculated from volts/div) and the extreme left and right grid lines (calculated from the timebase).
6. Choose a line style for your traces: solid **Line** or a disconnected series of sample **Points**.
7. To highlight more frequent samples, touch **Intensity** and enter a value from 0 to 100. For more information, see Adjust and Intensity.
8. If you selected to display an XY grid, select the source channels to **Input X** and **Input Y**.
9. If you have an external monitor installed, touch **Open Monitor Control Panel** and set up the external display.

Moving Traces from Grid to Grid

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

Next Grid Shortcut Button

Open the Channel setup dialog for the trace you want to move, then touch the **Next Grid** shortcut 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.

XY Displays

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

The sources must have the same X-axis scale.

1. Set up the desired source traces.
2. Go to **Display > Display Setup** and choose:
   - **XY** to display only the XY plot.
   - **XYSingle** to display the XY plot next to a single grid containing both source traces.
   - **XYDual** to display the XY plot next to two grids, each containing one of the source traces.
3. Touch **Input X** and **Input Y** and select your sources from the pop-up menu.

   NOTE: The inputs can be any combination of channels, math functions, or memories.

Persistence Overview

The Persistence feature displays waveforms in a manner that helps reveal idiosyncrasies or anomalies in a repetitive signal. Use Persistence to accumulate on-screen points from many acquisitions to see your signal change over time. The instrument persistence modes show the most frequent signal path in three-dimensional intensities of the same color, or graded in a spectrum of colors.

You can show persistence for up to eight inputs for any channel, math function, or memory location (M1 to M4).

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.

ANALOG MODE

When you select **Analog** mode, each trace is assigned a different color.

As a persistence data map develops, different intensities of that 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.

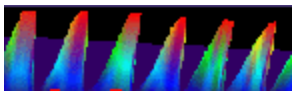
Color Mode

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

3d 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.

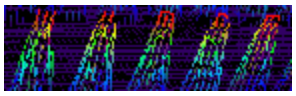
In this mode, you can also turn the X and Y axes of the waveform through 180° of rotation from -90° to +90°.



In the **solid** view of color-graded persistence, saturation is set at 50%, with red areas indicating highest intensity. The X-axis has been rotated 60%; the Y-axis has been rotated 15%.

In the **monochrome** (analog) view, the lightest areas indicate highest intensity, corresponding to the red areas in the solid view.

The **shaded** (projected light) view emphasizes the shape of the pulses.



In the **wire frame** view, lines of equal intensity are used to construct the persistence map.

Saturation Level

Besides the different modes, you can select a saturation level as a percentage of the maximum population. All populations above the saturation population are then assigned the highest color intensity: that is, they are saturated. At the same time, all populations below the saturation level are assigned the remaining intensities. Data populations are dynamically updated as data from new acquisitions is accumulated.

A saturation level of 100% spreads the intensity variation across the entire distribution; at lower saturation levels the intensity will saturate (become brighter) at the percentage value specified. Lowering this percentage causes the pixels to be saturated at a lower population and makes visible those events rarely seen at higher saturation levels.

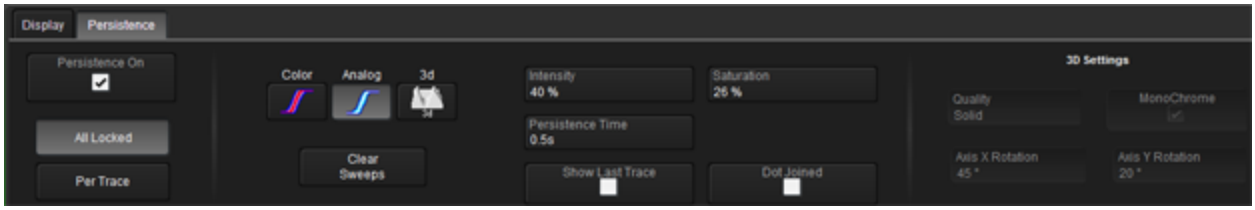
Persistence Time

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

Persistence Setup

This procedure explains how to set up the persistence display on traces. Persistence can be quickly added to all traces or applied individually.

Turn On Persistence



1. Access the Persistence dialog by choosing **Display > Persistence Setup**and touching the **Persistence tab**.
2. Check **Persistence On**.
3. To set up all traces together, touch **All Locked**.

   To set up traces individually, touch **Per Trace**.
4. Select the persistence mode: **None**, **Color**, **Analog**, or **3d**

   If you're doing individual setup, repeat the mode selection for each channel, then go to that channel's tab and make the remaining settings.
5. If using Analog or Color mode, optionally check **Show last trace** to superimpose the channel trace over the persistence display.

   If using 3d mode, complete setup for 3-D persistence.
6. Optionally, also change Saturation level and Persistence Time, and enable/disable **Dot Joined**.

Set Up 3-D Persistence

1. Touch the **3d** button and, if necessary, open the channel tab.
2. Under **3D settings**, touch **Quality** and choose **wire frame**, **solid**, or **shaded**.
3. Check **MonoChrome** if you prefer a single-color representation. In this case, intensity will be used instead of color to indicate more frequently occurring events.
4. Optionally, change the angle of display by entering new **Axis X Rotation** and **Axis Y Rotation** values from -90° to +90°.

   TIP: A quick way to rotate the display is to grab a corner and drag it in the desired direction.

Turn Off Persistence

To turn off the persistence display, access the Persistence dialog and choose **Reset All,** or select an individual channel's **None**(left-most) persistence mode button.

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 five, standard cursor types available.

The cursor measurement values can be read on the descriptor box for the trace. The **Show** buttons let you change which set of values are shown on the descriptor box. The available selections depend on the type of cursor.

The easiest way to position cursors is to touch and drag them to the desired locations. Use the Front Panel **Cursors knob** or the **Position** data entry controls at the right side of the Standard Cursors dialog to place the cursors precisely.

Cursor Types

Standard Cursors

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

Horizontal (Time) cursors place vertical lines through a desired point along the horizontal axis to read the signal's amplitude at the selected time. There are two main types:

- **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 place horizontal lines through a point on the vertical axis to read the amplitude of the signal at that point. The two types are:

- **Vertical Abs** - displays a single 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.

An option exists to place **Both** Horizontal (Time) and Vertical (Amplitude) types at once.

Special Cursors

Some cursors are offered only in special circumstances:

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

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

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

Display Cursors

Use either of the following methods to quickly turn on/off cursors:

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

Position Cursors

With the cursor on, turn the Front Panel **Cursors** knob. If there is more than one cursor line, push the Cursor knob until the correct line is selected, then turn the knob to move it.

OR

Touch and drag the cursor line to a new position.

Standard Cursors Dialog

These controls can be used in lieu of the Front Panel controls to set cursors. 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** - 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 1/2** - positive or negative time from the zero point.
- **Y 1/2** - number of positive or negative divisions from the zero level. May be a fraction of a division.
- **Track** - locks both cursor lines so that 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 like rise-time, rms voltage, and peak-to-peak voltage, for example.

The oscilloscope offers a selection of:

- Standard parameters for measuring amplitude and time
- Custom parameters you configure
- Specialized parameters for applications such as pass/fail testing or serial data decode (if you have those options installed)

You can configure and display up to eight measurement parameters at once.

Measure Gate

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

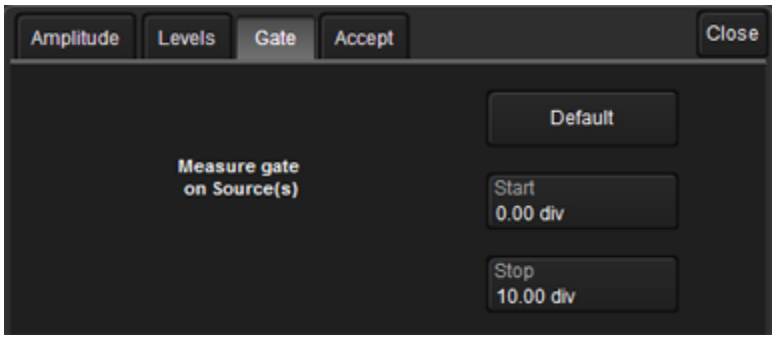
The default starting positions of the gate posts are 0 div and 10 div, which coincide with the left and right ends of the grid. The gate, therefore, initially encloses the entire waveform.

The quickest way to set a gate is by dragging the gate posts located at the far left and right of the grid to the desired positions.

You can refine this setting by specifying a position down to hundredths of a division in the **Gate Start** and **Stop** fields on the Gate right-hand dialog.

For Standard Horizontal or Standard Vertical parameters, all parameters share the same gate.

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



Level and Slope

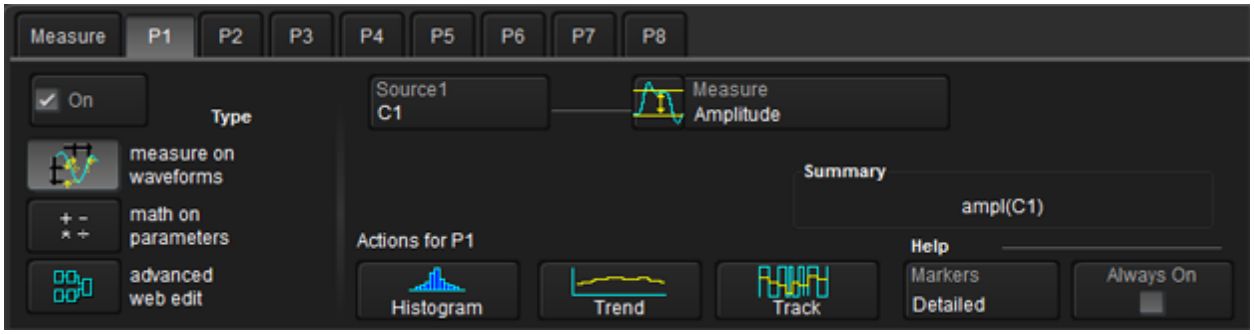
For several time-based measurements, you can choose to begin the measurement on positive, negative, or both slopes. For two-input parameters, such as Dtime@level, you can specify the slope for each input, as well as the level and type (percent or absolute).

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

Set Up Measurement Parameter

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

1. From the menu bar, choose **Measure > Measure Setup**.
2. Choose Measure Mode **My Measure**.
3. Touch the **Pxtab or button** of an unused location (or one that you want to change).



4. Select a **Type**:
   - **Measure On Waveforms**- measures directly on the waveform selected as Source1.
   - **Math On Parameters** - performs math (addition, subtraction, multiplication, division) on the parameters selected as Source1 and Source2. These must be two other custom parameters you have or will configure and saved to those slots.
   - **Advanced Web Edit** - uses Teledyne LeCroy's Processing Web for measurement setup. This feature, available with the XWEB option, allows you to chain practically unlimited math functions for operation on your waveform measurements.
5. Touch **Source1** and select the channel, math trace, memory trace, or other waveform to be measured.

   If using Math on Parameters, choose the parameters, rather than the source trace, in Source1 and Source2.
6. If you selected **Measure On Waveforms**, touch the **Measure** field and select the parameter 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.
8. Optionally, set a measurement gate by dragging the gate posts to reposition them or by entering a **Start** and **Stop** division on the Gate right-hand dialog..
9. Check **On** to enable the parameter and add it to the measurement readout table.
10. Check **Show Table** to display the readout on screen.

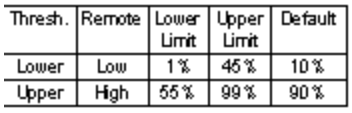
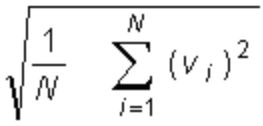
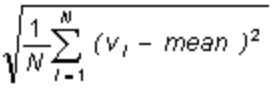
List of Standard Parameters

Standard measurement parameters are listed below alphabetically.

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

Parameter	Description
Amplitude (ampl)	Measures the difference between upper and lower levels in two-level signals. Differs from pkpk in that noise, overshoot, undershoot, and ringing do not affect the measurement. Amplitude is calculated by using the formula Top – Base. On signals not having two major levels (such as triangle or saw-tooth waves), the amplitude parameter returns the same value as peak-to-peak.
Area	Integral of data: Computes area of the waveform relative to zero level. Values greater than zero contribute positively to the area; values less than zero, negatively.
Base	Lower of two most probable states (higher is top). Measures lower level in two-level signals. Differs from min in that noise, overshoot, undershoot, and ringing do not affect measurement. On signals not having two major levels (such as triangle or saw-tooth waves), the amplitude parameter returns the same value as minimum.
Cycles (cycles)	Determines number of cycles of a periodic waveform lying between cursors. First cycle begins at first transition after the left cursor. Transition may be positive- or negative-going.
Delay	Time from trigger to transition: Measures time between trigger and first 50% crossing of specifies signal. Delay can be used to measure the propagation delay between two signals by triggering on one and determining delay of other.
Delta Delay (ddelay)	Computes time between 50% level of two sources.
Dperiod@level (dper@lv)	Adjacent cycle deviation (cycle-to-cycle jitter) of the period measurement for each cycle in a waveform. The reference level for this measurement can be specified.
Dtime@level (dt@lv)	Computes the time between transitions of the selected sources at the specified levels. Only positive going transitions are counted.
Dtrig Time (dtrig)	Time from last trigger to this trigger
Duration (dur)	For single sweep waveforms, dur is 0; for sequence waveforms: time from first to last segment's trigger; for single segments of sequence waveforms: time from previous segment's to current segment's trigger; for waveforms produced by a history function: time from first to last accumulated waveform's trigger.
Duty Cycle	Percent of period for which data are above or below the 50% level of the signal.
Duty@level (duty@lv)	Percent of period for which data are above or below a specified level.
Edge@level (edge@lv)	Number of positive edges in waveform that cross the specified threshold level.
Fall 80-20% (fall8020)	Duration of pulse waveform's falling transition from 80% to 20% of the amplitude averaged for all falling transitions between the measurement gates. On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving less predictable results.
Fall time (fall)	Duration of pulse waveform's falling transition from 90% to 10% of the amplitude averaged for all falling transitions between the measurement gates. On signals not having two major levels (triangle or saw-tooth waves, for example), top and base can default to maximum and minimum, giving less predictable results.
Fall@level (fall@lv)	Fall at level: Duration of pulse waveform's falling edges between user-specified transition levels.

Parameter	Description
	<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. Formulas for upper and lower values: lower = lower thresh. x amp/100 + base upper = upper thresh. x amp/100 + base
First	Indicates value of horizontal axis at left cursor.
Frequency (freq)	Period of cyclic signal measured as time between every other pair of 50% crossings. Starting with first transition after left measurement gate. The period is measured for each transition pair. The reciprocal of each period measurement is calculated as the frequency.
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.
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.
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 point** checkbox is checked, it gives the vertical value of the nearest data point.
MATLAB	Produces a parameter using a user-specified MATLAB function.
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.
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.
Peaks	Number of peaks in a histogram.
Peak to Peak	Difference between highest and lowest points in waveform. Unlike ampl, does not assume the waveform

Parameter	Description
(pkpk)	has two levels. Peak to peak is calculated using the formula *maximum – minimum*.
Percentile (pctl)	Horizontal data value that divides a histogram so the population to the left is xx% of the total.
Period	The time between every other pair of 50% crossings. Starting with first transition after left measurement gate, period is measured for each transition pair, with values averaged to give final result.
Period@level (per@lv)	The time between every other pair of at the level specified. Starting with first transition after left measurement gate, period is measured for each transition pair, with values averaged to give final result.
Phase	Phase difference between signal analyzed and signal used as reference. Both signals are measured from the 50% point of their rising edges.
Rise	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 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@level (rise@lv)	Rise at level: 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. Formulas for upper and lower values: lower = lower thresh. x amp/100 + base upper = upper thresh. x amp/100 + base
RMS	Root Mean Square of data between the measure 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 clock1 edge minus time of nearest clock2 edge. Both signals are measured from the 50% point of their rising edges.
Slew Rate (slew)	Slew rate or local dV/dt in a transition zone
Std Dev (sdev)	Standard deviation of the data between the measure gates using the formula:  Where: vi denotes measured sample values, and N = number of data points within the periods found up to maximum of 100 periods. This is equivalent to the rms for a zero-mean waveform. Also referred to as AC RMS

Thresh.	Remote	Lower Limit	Upper Limit	Default
Lower	Low	1%	45%	10%
Upper	High	55%	99%	90%

Parameter	Description
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.

Quick Measurements

Once you have set custom parameters in an available location, you can quickly hide or display them by choosing **Measure > My Measure** and checking **On** next to each parameter you want to display. You do not have to go through the entire setup process.

There are also standard parameter sets 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 Horizontal** for a full set of common voltage parameters: mean, sdev, max., min., ampl, pkpk, top, base.

Mark the **Show Table** checkbox to display the parameter readout table below the grid.

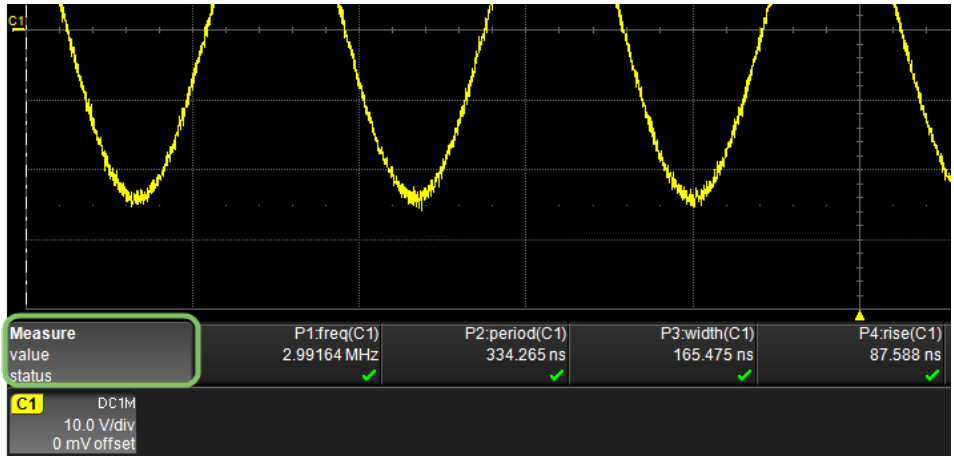


Table of measurements open beneath grid. Far left cell opens the Measure dialog.

To quickly access the Measure Setup dialog if it is closed, touch the far left cell of the readout table labeled Measure.

View Statistics

You can add the statistical measures **value(last)**, **mean**, **min.**, **max.**, **sdev**, and **num**(ber of measurements computed) to the measurement parameter readout table.

To turn on statistics, access the Measure dialog and check **Statistics On**. Clear the checkbox to remove statistics from the readout. You can also choose **Measure > Statistics** from the menu bar.

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

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

To reset the statistics counter, touch **Clear Sweeps** on the display or Front Panel.

View Histicon

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.

1. Choose **Measure > Measure Setup** from the menu bar to access the Measure dialog.
2. Select **Show Table**.
3. Check **On** to enable the parameters you wish to display.
4. Select **Statistics Histicons**.

NOTE: You can quickly display a full histogram by touching the histicon you want to enlarge. The enlarged histogram appears superimposed over its source trace.

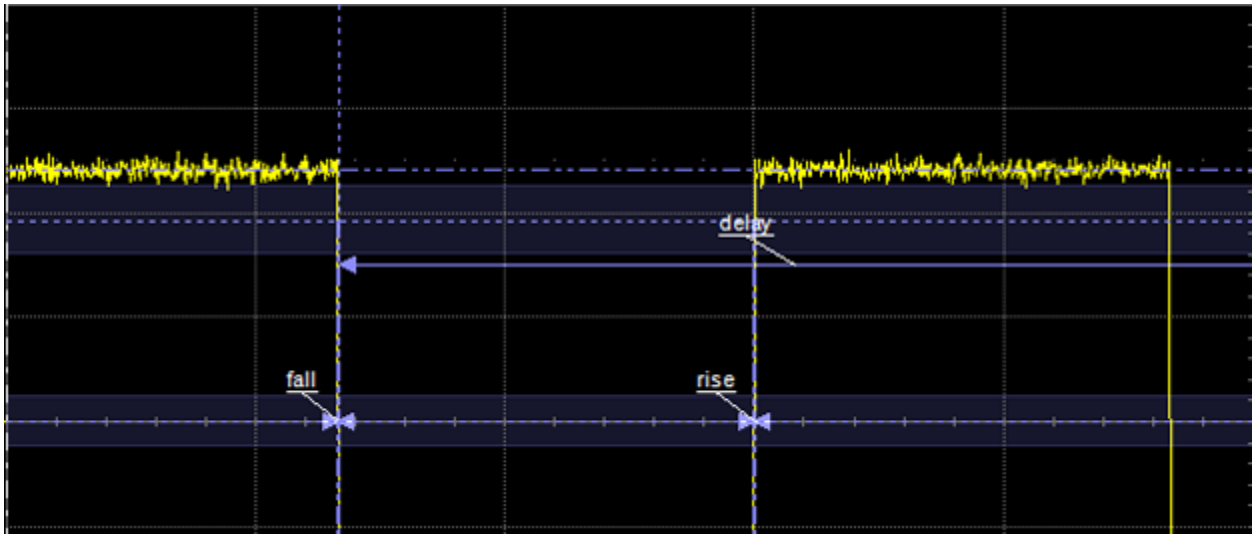
Help Markers

Help Markers clarify measurements by displaying cursor lines and labels marking the points being measured. 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.

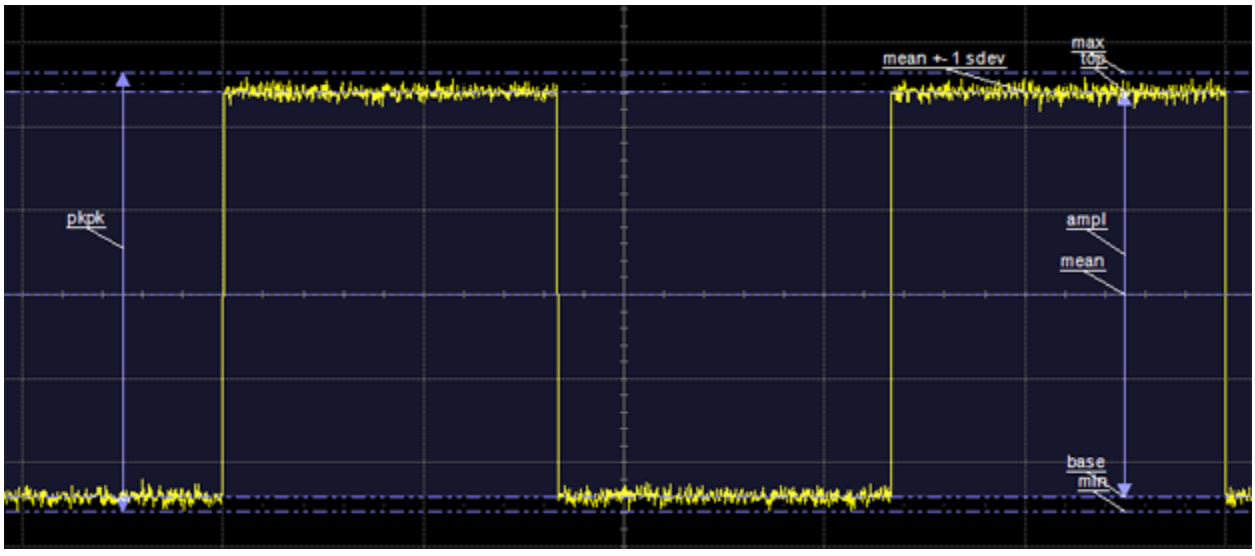
You can choose to use **Simple** markers, which are only the lines, or **Detailed** markers, which include the measurement point labels.

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 Help 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.



Standard Horizontal Parameter Help Markers



Standard Vertical Parameter Help Markers

Turn On Markers

1. From the menu bar, choose **Measure > Measure Setup**.
2. Select a Measure Mode: **Std Vertical**, **Std Horizontal**, or **My Measure**.
3. Touch the **Show All** button to display Help Markers for each enabled parameter.

   The type of markers last selected appear on the display.

   NOTE: If you choose My Measure but have not yet set up or enabled any parameters, you will not see any markers, either.
4. To change the marker type, open any parameter (Px) dialog and in **Markers** select either:
   - **Simple** - produces cursors and gate posts. The gate posts are independently placeable for each parameter.
   - **Detailed** - produces cursors, gate posts, a label identifying the parameter being measured, and a level indicator and hysteresis band for "at level" parameters.

   NOTE: The Markers setting is applied to all parameters at the same time. If you choose Simple markers on any parameter dialog, all parameters are then displayed in this mode.
5. Select the **Always On** checkbox if you wish to continue to display Help Markers on open traces.

Turn Off Markers

From the **Measure setup dialog**, choose Help Markers **Clear All**.

From any **Px dialog**, choose **Markers Off**.

Qualified Parameters

Some Teledyne LeCroy software packages give you the ability to constrain parameter measurements to a vertically or horizontally limited range, or to occurrences gated by a second waveform. Furthermore, both constraints can operate together. This capability enables you to exclude unwanted characteristics from your measurements. It is much more restrictive than See "Measure Gate" on page 55 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.

Range Limited Parameters

1. From the menu bar, choose **Measure → Measure Setup...**.
2. Touch a **Px** tab to open its corresponding dialog.
3. Now, on the dialog, touch inside the **Source** control and select a source from the pop-up.
4. Touch inside the **Measure** control and select a parameter from the pop-up menu.
5. Touch the **Accept** tab of the dialog on the right, then touch the **Values In Range** checkbox.

   NOTE: Depending on whether you select a vertical or horizontal parameter, the correct units will be automatically displayed (V, s, Hz, dB) in the Between and And controls. Or, if you select a simple ratio parameter (such as power factor) that yields a dimensionless number, no units will be displayed.
6. Touch the **Find Range** button to quickly display the most recent value of the parameter measurement.

Waveform Gated Parameters

1. From the menu bar, choose **Measure → Measure Setup**.
2. Touch any **P*x*** tab to open the setup dialog.
3. Touch **Source** and select a source from the pop-up menu.
4. Touch **Measure** and select a parameter from the pop-up menu.
5. Touch the **Accept**righ-hand dialog tab, then check the **Values Based on Waveform State** box.
6. Touch **When Wform** and select the gating source.
7. Touch **State Is** and select **High** or **Low** from the pop-up menu. Parameter measurements on the subject waveform will only be taken when the gating waveform is in the selected state.
8. Touch**Level Type** and select **Absolute** or **Percent** from the pop-up menu.
9. Touch**Level** and enter the crossing level value at which you want measurements to begin..

   You can instead touch the **Find Level** button to automatically select the 50% level of your gating waveform.

Math on Parameters

Besides reading parameter measurements, you can set up a custom parameter that performs arithmetic operations (addition, subtraction, multiplication, division) on two other parameter measurements. Alternatively, you can apply mathematical functions (for example, invert) to a single parameter measurement.

The setup for Math on Parameters is much like other custom parameter setup. The only significant difference is the choice of Math on Parameters instead of Measure on Waveforms and the selection of source parameters instead of source traces. There is added functionality for using custom scripts to calculate the results.

Math on Parameters differs from Math in that the input and the output are still numerical values, as are all parameter measurements. Math functions, on the other hand, input and output waveform traces. Math on Parameters results display in the parameter readout table.

Exclusions

Logarithmic Parameters

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

- auto-correlation signal-to-noise ratio (ACSN)
- narrow-band power (NBPW)
- media signal-to-noise ratio (MSNR)
- residual signal-to-noise ratio (RSNR)
- top-to-base ratio when the units are in dB (TBR)

Other Excluded Parameters

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

- delta clock-to-data near (DC2D)
- delta clock-to-data next (DC2DPOS)
- delta clock-to-data previous (DC2DNEG)
- delta delay (DDLY)
- delta time at level (DTLEV)
- phase (PHASE)
- resolution (RES)
- mTnTmT shift (BEES)
- mTnTmT shift sigma (BEESS)
- mTnTmT shift sigma – list (BEESS)

Set Up Math on Parameters

1. Touch **Measure → Measure Setup...** on the menu bar.
2. Choose Measure Mode **My Measure**.
3. Touch output **Px tab or button** to display the parameter setup dialog.
4. Touch the **Math on Parameters** button.
5. Touch **Math Operator** and select a math operation from the **Select Measurement** menu.

   If you select an operation that requires two input parameters, the **Source1** field will expand to two fields.
6. Touch **Source1** and **Source2** and select two input parameters (P1 to P8) other than the parameter you are now setting up.

   To apply math to a single parameter (for example, Invert), just select it in **Source1**.
7. Check **On** to enable the new output parameter and add it to the display.

Using Scripts

In addition to the arithmetic operations, you can write your own VBScript or JavaScript to apply to one or two measurement parameters. When setting up the output parameter, choose the Math Operator **P Script**.

Scripting can be done in the **Script Editor** window directly on the instrument, or you can import an existing script.

PARAM SCRIPT VS. P SCRIPT

Param Script is a VBScript or JavaScript that operates on one or two *waveforms* and outputs a parameter measurement, as shown in the figure below. P Script, on the other hand, is another VBScript or JavaScript that takes as input one or two *parameters* and performs a math operation on them to produce another parameter output.

The inputs to Param Script can also be math (Fx) or memory (Mx) traces. The inputs to P Script can be the results of any parameter measurement, not necessarily Param Script.

Set Up Math On Parameters Using Script

1. Touch **Measure → My Measure...** on the menu bar.
2. Touch the output **Px tab or button** to display the parameter setup dialog.
3. Touch the **Math on Parameters** button.
4. Touch **Source1** and **Source2** and select the input parameters (P1 to P8).

   If you are applying math to a single parameter (for example, invert), just select it in Source1.
5. Touch **Math Operator** and choose **P Script** from the **Select Measurement** menu.
6. In the right-hand **Script Math** dialog, touch **Script Language** and choose either **VBScript** or **JScript**.
7. Touch the **Edit Code** button.

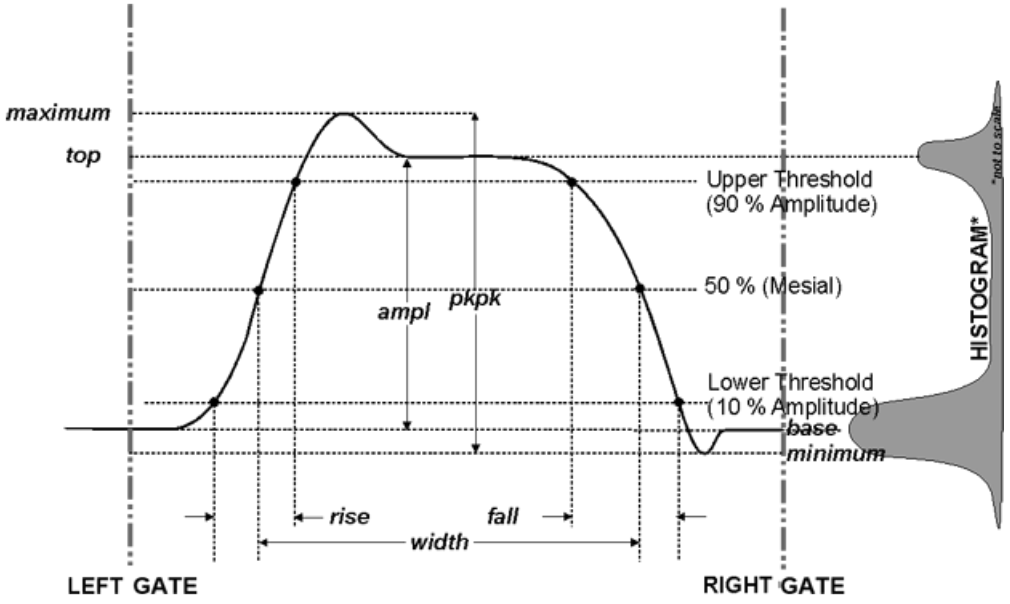
   The **Script Editor** window opens.
8. Enter code in the script editor, or call up an existing script from a file storage location.

   If you create your script in this window, you can export it to a new file.

Calculating Measurements

Determining Top and Base Lines

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



Determining Rise and Fall Times

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

Threshold levels for rise or fall time can also be selected using absolute or relative settings (r@level, f@level) if these parameters are included in your oscilloscope. 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

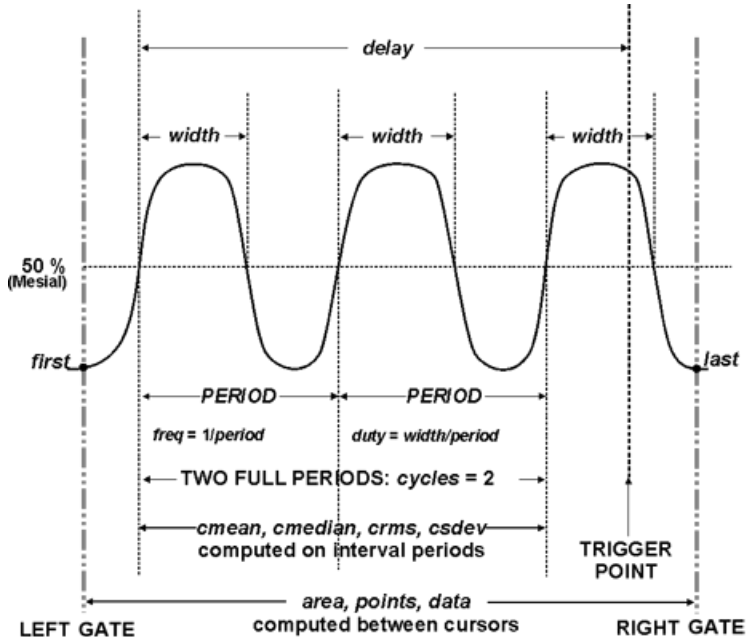
$$\frac{1}{Mf}\sum_{i=1}^{Mf}(Tf_i^{10} - Tf_i^{90})$$

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

Determining Time Parameters

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

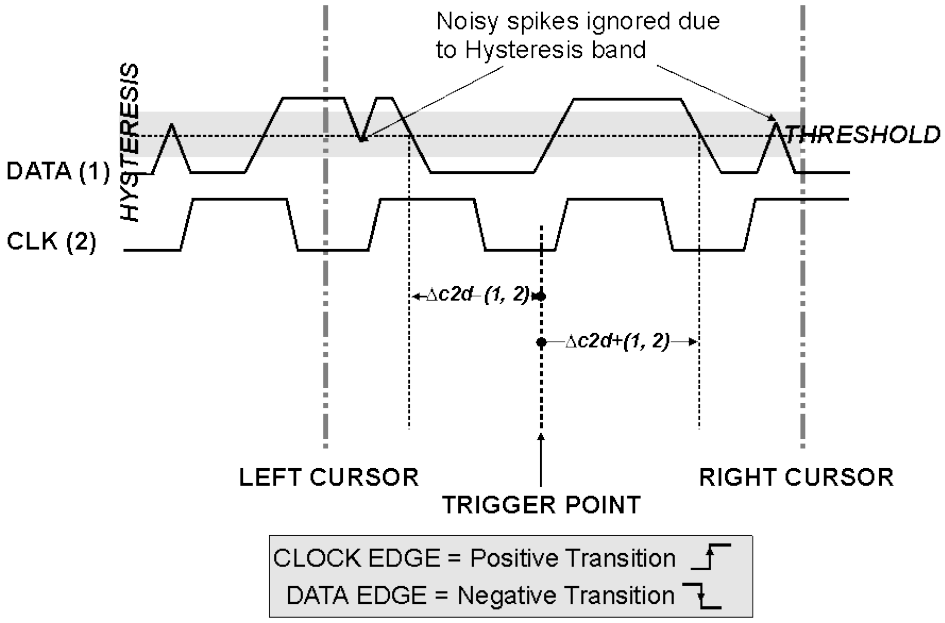
Time-parameter estimation depends on the number of cycles included within the observation window. If the number of cycles is not an integer, parameter measurements such as rms or mean will be biased. However, only the last value is actually displayed, the mean being available when statistics are enabled.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

If included in your oscilloscope, parameters such as Delta c2d± require the transition polarity of the clock and data signals to be specified.



Moreover, a hysteresis range may be specified to ignore any spurious transition that does not exceed the boundaries of the hysteresis interval. In the figure, Delta c2d- (1, 2) measures the time interval separating the rising edge of the clock (trigger) from the first negative transition of the data signal. Similarly, Delta c2d+ (1, 2) measures the time interval between the trigger and the next transition of the data signal.

Math

Teledyne LeCroy offers a deep and always growing toolset of math functions.

Math functions can be applied to any channel (**C*x***), zoom (**Z*x***), memory (**M*x***), or even other math traces (**F*x***), allowing you to chain operations. For example, trace F2 can show the average of C1, while trace F3 provides the integral of F2.

In addition to the extensive math capabilities that are standard with every oscilloscope, enhanced math analysis tools customized for various industries and applications are offered through optional software packages. To learn about math tools available in each optional package, see the datasheets on the Teledyne LeCroy website at teledynelecroy.com. If you have installed software options, these capabilities are accessed through the oscilloscope Analysis menu, rather than the Math menu, although special measure parameters and math functions will be available when using Measure and Math dialogs.

Single vs. Dual Operation 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 together.

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):

Graphing

The **Graph button** on the Math Function (Fx) dialogs allows you to create math functions that plot the results of an applied measurement parameter: histogram, track, or trend. Choose the source, the measurement parameter, and the type of plot to draw. The plots are the same as those you would create using the shortcut buttons on the Measure Parameter (Px) dialog. See About Histograms and Track vs. Trend.

As with other math functions, any configurable settings will appear on right-hand dialogs, after the plot type is selected.

Set Up Math Function

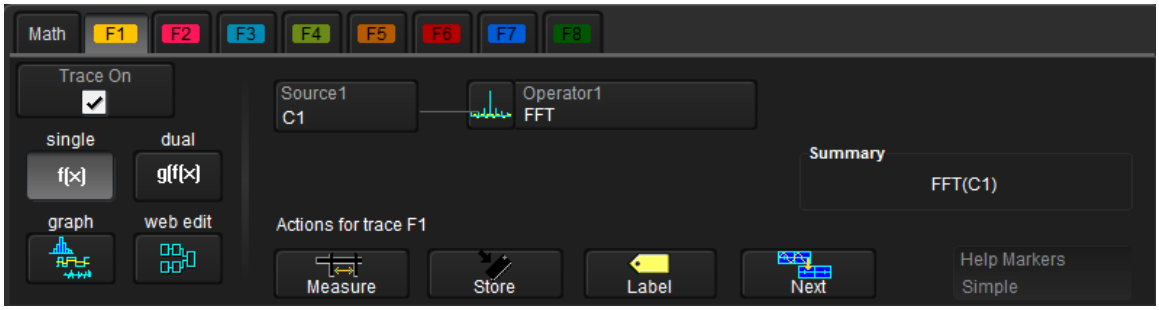
This procedure explains how to set up single or dual operator math function (F*x*) traces. Function traces take as input one or more channel, zoom, memory or math traces and output a new math trace.

For more information about creating math traces that plot the results of applied measurements, see View Trend, View Track, and View Histogram.

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

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

2. Choose a location by touching one of the **F*x* tabs** (F1-F8).



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

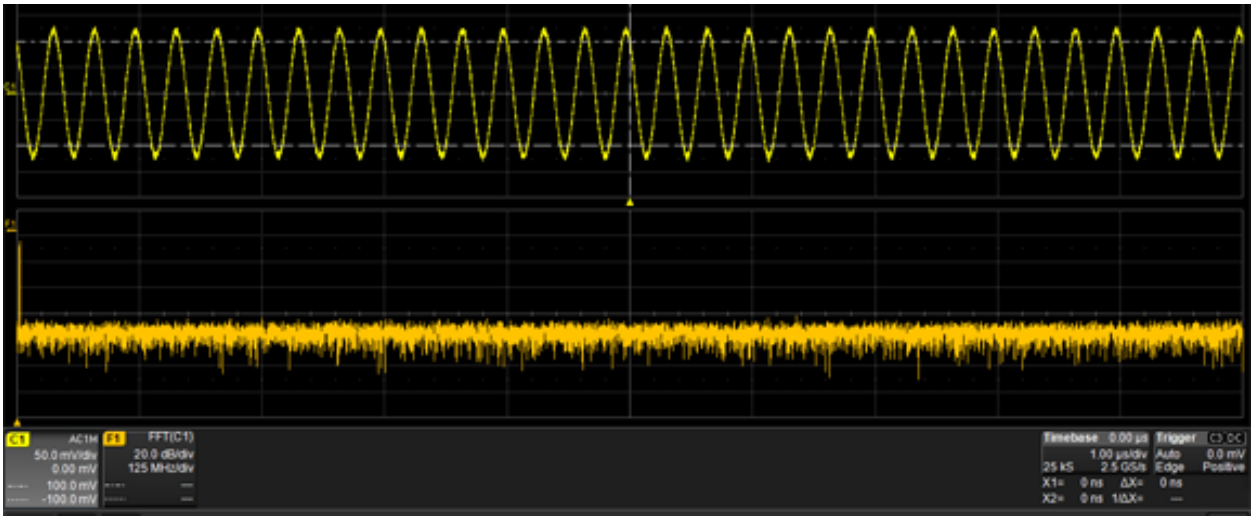
   A **Summary** of the function you are building appears on the dialog. Refer to this to be sure your sources are in the proper order to yield the function you want (e.g., C1-C2 vs. C2-C1).

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 rescale the math trace. This does not affect the scale of any other traces.

7. If you're creating a dual function, repeat Steps 4 through 6 for the second operation.

8. Check **Trace On** to display the new math trace.



Enable/Disable Math Function

Once a math function has been created and saved in one of the F*x* locations, 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.

Clear the On box to disable the function and close the trace.

List of Math Functions

Standard math functions are listed below alphabetically.

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

Function	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.
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.

Function	Definition
DigitalAND	AND function between two digital waveforms (-MS models only).
DigitalFlipFlop	Input1 is clocked in a hold when a rising edge of input2 occurs (-MS models only).
DigitalNAND	NAND function between two digital waveforms (-MS models only).
DigitalNOR	NOR function between two digital waveforms (-MS models only).
DigitalNOT	NOT function (inverter) of a digital waveform (-MS models only).
DigitalOR	OR function between two digital waveforms (-MS models only).
DigitalXOR	XOR function between two digital waveforms (-MS models only).
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.
Histogram	Plots the number of data points that fall into statistically significant intervals or bins. Bar height relates to the frequency at which data points fall into each interval/bin.
Integral	Calculates the linearly rescaled integral (with multiplier and adder) of a waveform input starting from the left edge of the screen using the formula: *(current sample value + next sample value) * (horizontal sample interval)* Each calculated area is summed with the previous sum of areas. The multiplier and adder are applied before the integration function.
Interpolate	Inserts points between sampled points (upsamples) according to one of three algorithms: Linear (straight line), Sinx/x (curved), and Cubic (spine). Interpolation factor of 2 to 50 determines number of points in the upsample.
Invert	For every point in the waveform, the inverse of that point is calculated.
Ln	Peforms a natural log of a waveform. Values less than or equal to zero are set to underflow.
Log10	Performs a log base 10 of a waveform. Values less than or equal to zero are set to underflow.
MatLab math	Applies a pre-programmed MatLab math function to the source waveform. Requires XDEV option to edit functions through the oscilloscope GUI using MatLab Script.
phistogram	Creates a 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.

Function	Definition
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.
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.
Trk	Same as Track, with alternate transition types.
Trend	Produces a waveform composed of a series of parameter measurements in the order the measurements were taken. The vertical units are those of the source parameter, the horizontal unit is measurement number. The trend contains a single value for each measurement. See View Trend.
Zoom	Produces a magnified trace of a selected portion of the input waveform. See Zooming Traces.

Interpolation

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. Touch the **Interpolate** tab in the mini setup dialog to the right of the main dialog.
3. Touch inside the **Algorithm** control and select an interpolation type.
4. Touch inside the **Upsample by** control (Upsampling is the factor by which sampling is increased) and enter a value.

Sparser 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, tells the oscilloscope to retain only one out of every 4 samples. A **Sparsing offset** of 3, on the other hand, tells the oscilloscope 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.

NOTE: The maximum sparsing offset that can be entered for any sparsing factor equals Sparsing Factor 1.

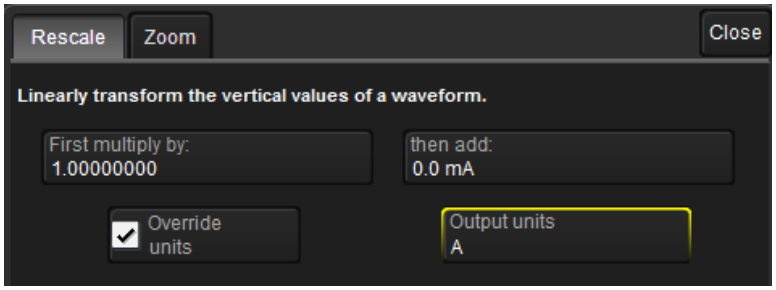
1. Follow the usual steps to set up a math function, selecting **Sparse** from the **Misc** submenu.
2. Touch the **Sparsing factor** control and provide a Bandwidth Limit value.
3. Touch the **Sparsing offset** control and provide a value.

Rescaling and Assigning Units

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

Set Up Rescaling

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



3. To apply a multiplication factor:
   - Check the **First multiply by:** box and enter a value for *a*, the multiplication factor.
   - Touch **then add:** and enter a value for *b*, the additive constant.
4. To change the output unit of measure from that of the source waveform:
   - Check **Override units**.
   - In **Output** enter the abbreviation for the unit the measure you wish to use.

   You can also enter combinations of the unit abbreviations following these rules:
   - For the quotient of two units, use the character "**:/**"
   - For the product of two units, use the character "**.**"
   - For exponents, append the digit to the unit without a space: S2 = seconds squared.

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

Abbreviated Units of Measure

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

Enhanced Resolution

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

Use ERes:

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

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

Set Up Enhanced Resolution (ERes)

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

To apply ERes as a Math function:

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

How the Instrument Enhances Resolution

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

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

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

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

The improvement in SNR corresponds to the improvement in resolution if the noise in the signal is white (evenly distributed across the frequency spectrum).

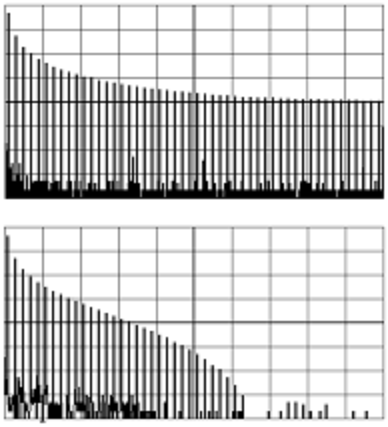
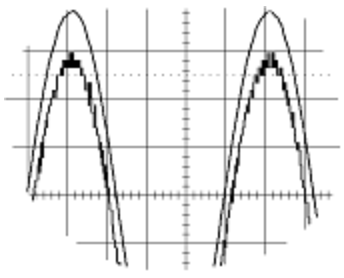
If the noise power is biased towards high frequencies, the SNR improvement will be better than the resolution improvement.

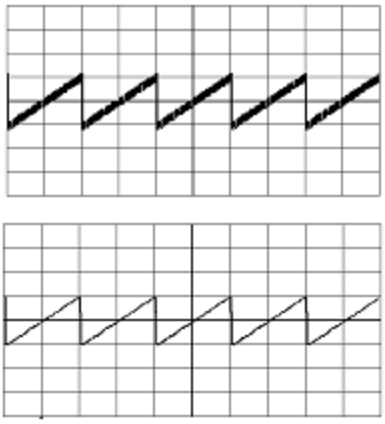
The opposite may be true if the noise is mostly at lower frequencies. SNR improvement due to the removal of coherent noise signals - feed-through of clock signals, for example - is determined by the fall of the dominant frequency components of the signal in the passband. This is easily ascertained using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

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

Example ERes Applications

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

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

Graph	Function
	**To reduce noise:** The example at left shows enhanced resolution of a noisy signal. The original trace (left top) has been processed by a 2-bit enhanced resolution filter. The result (left bottom) shows a smooth trace, where most of the noise has been eliminated.

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

Averaging Waveforms

Summed Averaging

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

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

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

Continuous Averaging

NOTE: Continuous Averaging may be set up from either the Channel dialog under Pre-Processing, 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 gradually 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 continuous averaging is:

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

This is also the formula used to compute summed averaging. But by setting a "sweeps" value, you establish a fixed weight that is assigned to the old average once the number of "sweeps" is reached. For example, for a sweeps (weight) value of **4**:

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

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

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

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

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

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

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

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

NOTE: The number of sweeps used to compute the average will be displayed in the bottom line of the trace descriptor label:

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.

FFT

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

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

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

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

Set Up FFT

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

3. Choose to either:
   - **trunc(ate)** - When the FFT transform size does not match the record length, truncate the record and perform an FFT on the shorter record. This option increases the resolution bandwidth.
   - **zero-fill** - When the source data for the FFT comes from a math operation that shortens the record (as is commonly encountered in filtering operations like ERes), replace the missing data points wityh data values whose amplitudes are interpolated to fit between the last data point and the first data point in the record. This guarantees that there is not a first-order discontinuity in the filled data. Since the data at the end of the record is filled data, it is advisable to select a weighting window other than rectangular to minimize the effect of the fill on the resulting spectrum.
4. Check the **Suppress DC** box to make the DC bin go to zero. Otherwise, leave it unchecked.
5. Choose an **Output type**.
6. Optionally, choose a weighting **Window**. See the section below for more information about FFT weighting windows.
7. Touch **Algorithm** and choose either:

- **Least Prime** (default) - a least primes algorithm that computes FFTs on transform sizes having lengths that can be expressed as factors of 2^N*5^K. This is very compatible with the record lengths encountered in the oscilloscope, which are often multiples of 1, 2, 4, 5, or 10.
- **Power of 2** - a power of 2 algorithm where the record lengths are in the form of 2^N. The power of 2 algorithm generally runs faster than the least primes algorithm. The price that is paid is a record length that is not the same as the acquired signal. The power of 2 FFT truncates to the nearest power of 2 less than record length (if truncate is chosen) or fill data to nearest power of 2 greater than the record length (if zero fill is selected).

8. Depending on your **Output Type** selection, you may also make selections for :
   - **Group Delay Shift**
   - **Line Impedence** - by default, the FFT function assumes that the oscilloscope is terminated in 50 Ohms. If an external terminator is being used, this setting can be changed to properly calculate the FFT based on the new termination value.

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 thus 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. Alternative functions with less attenuation (Flat Top and Blackman-Harris) provide maximum amplitude at the expense of frequency resolution. Whereas, Hamming and Von Hann are good for general purpose use with continuous waveforms.

Window Type	Applications and Limitations
Rectangular	These are normally used when the signal is transient (completely contained in the time-domain window) or known to have a fundamental frequency component that is an integer multiple of the fundamental frequency of the window. Signals other than these types will show varying amounts of spectral leakage and scallop loss, which can be corrected by selecting another type of window.
Hanning (Von Hann)	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Hamming	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Flat Top	This window provides excellent amplitude accuracy with moderate reduction of leakage, but with reduced frequency resolution.
Blackman-Harris	It reduces the leakage to a minimum, but with reduced frequency resolution.

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

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. That is, 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, you can optionally **Reset Count** or **Change BatchSize**.

Analysis

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

These measurements and manipulations of the original input signal 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. See About Histograms.
- **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. See View Track.
- **Trends** provide a view of a measurement parameter over an extended period of time and over multiple acquisitions. See View Trend and Track vs. Trend to better understand what a Trend provides compared to a Track.

There are also conditional tests that can be applied to the data to find particular events:

- Pass/Fail Testing, including Pass/Fail Testing, finds normal/abnormal measurements as indicated by whether or not they meet a set of defined criteria.
- WaveScan searches a single acquisition for events that meet specific criteria, enabling you to zoom in on anomalies in the waveform, or scans multiple acquisitions with allowable trigger actions when conditions are met. It can also be used to filter measurements. A variety of views help you understand the behavior of waveforms.

Finally, History Mode facilitates analysis by enabling you to quickly return the waveform display to any point in an acquisition history.

Optional software packages may be purchases that simplify specialized analysis, such as various Serial Data Decode options. These all add new methods to those available on the oscilloscope Analysis menu.

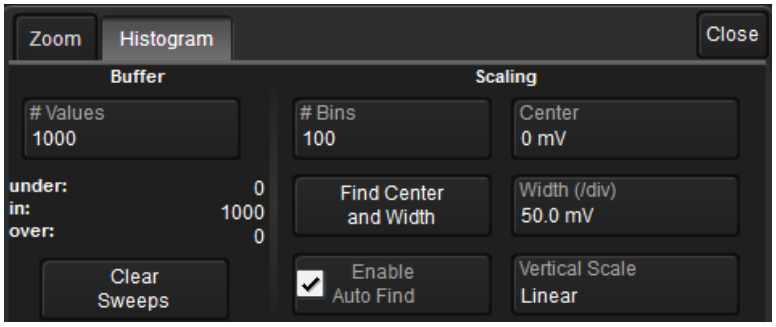
View Histogram

1. If you are not already on the Measure or Math dialog, choose **Measure → Measure Setup...** or **Math → Math Setup...**from the menu bar.
2. Touch the tab for the measurement parameter or math function you wish to histogramand check **Trace On**.
3. If you're already on the Fx dialog, touch the **graph button** and skip to Step 5.

   OR

   Touch the **Histogram** button at the bottom of the Px dialog and choose the math trace (**F1-F12**) in which to display the histogram.

   The histogram opens in a new grid along with its function descriptor box.
4. Touch the new **Fx descriptor box** to display the F*x* dialog.
5. Touch the **Histogram tab** at the right to display the Histogram right-hand dialog.



6. 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.
7. Touch **#Bins** and enter the number of bins that comprise the histogram. This determines how many bars appear in the histogram.
8. To let the oscilloscope determine the range of values represented by each bin/bar, check **Enable Auto Find**, then touch the **Find Center and Width button**.

   OR

   To set your own range, enter **Center** and **Width** values.

View Persistence Histogram

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

NOTE: This math 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 dialog, then touch **Source1** and select a source trace from the pop-up.
4. Touch the **Phistogram tab** at the right to open the Phistogram dialog.
5. Touch **Slice Direction** and select **Horizontal** or **Vertical** slice from the pop-up menu.
6. Touch **Slice Center** and use the pop-up keypad to enter a value.
7. Touch **Slice Width** and use the pop-up keypad to enter a value.

Track and Trend

Both Track and Trend are tools that can be used to plot measurement data and observe variations with respect to time. Differences between Track and Trend are summarized in the following table:

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 oscilloscope takes time to calculate measurement values and to display data before the trigger is re-armed, data can be missed.

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 of the oscilloscope between acquisitions. Think of Trend as a strip chart recorder for your oscilloscope.

View Track

This procedure explains how to view the Track of a measurement parameter applied to a waveform. A track is a waveform 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.

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

1. If not already on the Measurement dialog, choose **Measure → Measure Setup...**.
2. Touch the **Px** tab for the parameter you wish to plot.
3. Touch the **Track** button at the bottom of the Px dialog and select a math function (**Fx**) in which to draw the Track.

   The Track is displayed on a new grid, along with its function descriptor box.
4. To rescale the Track plot:
   - Touch the **Track function descriptor box** to open the Fx dialog, then touch the **Track tab**.

- On the Track right-hand dialog, **uncheck Auto Find Scale** and enter a new **Center** and **Height/div**.

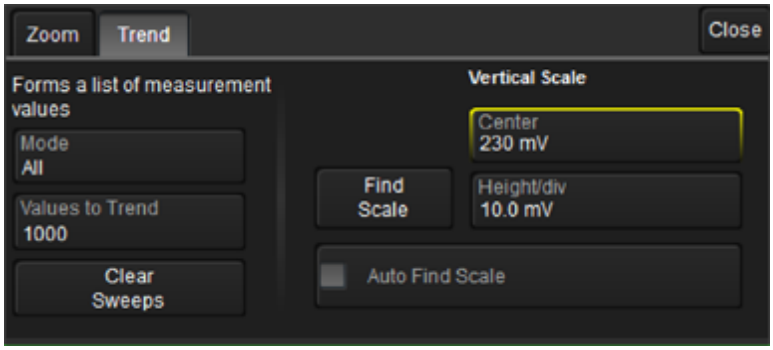
View Trend

This procedure explains how to view the trend of a measurement parameter. A trend is 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.

Although the trend plots measurement values, the plot is drawn as a math function and controlled through the Math dialog.

1. If you're not already on the Measure dialog, choose **Measure → Measure Setup...**.
2. Touch the **P*x* tab** for the parameter you wish to plot.
3. Touch the **Trend button**at the bottom of the dialog and choose a math function **F*x*** in which to draw the Trend.

   The Trend is displayed in a new grid, along with its function descriptor box.
4. To rescale the Trend plot:
   - Touch the **Trend function descriptor box** to open the Math dialog, then touch the **Trend tab** at the far right of the dialog.
   - On the Trend right-hand dialog, uncheck **Auto Find Scale** and enter the new **Center** and **Height** values.



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. The results are time stamped, tabulated, and can be selected for individual viewing.

There are two principal approaches to using WaveScan.

Capture & Search -- Make a single acquisition, then use Measurement Mode to search for parameter measurements that fit your filter criteria.

Scan -- Set up the scan mode, then scan for matching events across multiple acquisitions.

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

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

WaveScan window with all scan "views" turned on.

Scan Modes

The scan mode determines the type of search to be performed. Select the Mode along with the Source trace to be searched on the main WaveScan dialog.

For each mode, different controls appear on the WaveScan dialog, providing additional inputs to the search criteria. Make the appropriate entries in these fields before starting the search.

EDGE MODE

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

- **Slope** -- choose Pos, Neg, or Both.
- **Level is** -- choose Percent or Absolute.
- **Percent/Absolute Level** -- Enter a threshold value as a percentage of Top to Base or voltage level. A marker displayed over the source trace indicates the level.

NON-MONOTONIC MODE

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

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

RUNT MODE

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

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

- **Runt Type** -- choose Both, Pos, or Neg.
- **Hysteresis** -- enter the hysteresis level as a percentage or voltage.
- **Low Threshold** and **High Threshold** -- enter the levels as a percentage or voltage.
- **Absolute Levels** -- check this box if you want to enter levels as absolute voltage instead of percentage.

Measurement Mode

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

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

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

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. Additional settings for Serial Pattern Mode are:

- **Viewing** -- choose to enter the pattern as Binary or Hex.
- **Binary/Hex** -- enter the pattern.
- **Num. Patterns to detect** -- enter a whole number.

Bus Pattern Mode

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

- **Viewing** -- choose to enter the pattern as Binary or Hex.
- **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 displayed varies from one to three grids depending on which views are enabled. WaveScan handles this function automatically, and there is no option to move traces from one grid to another, as would be the case under normal operation.

You'll find additional controls for manipulating views like Scan Overlay and Zoom on their respective dialogs. If you turn on these traces from their 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 And Times

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

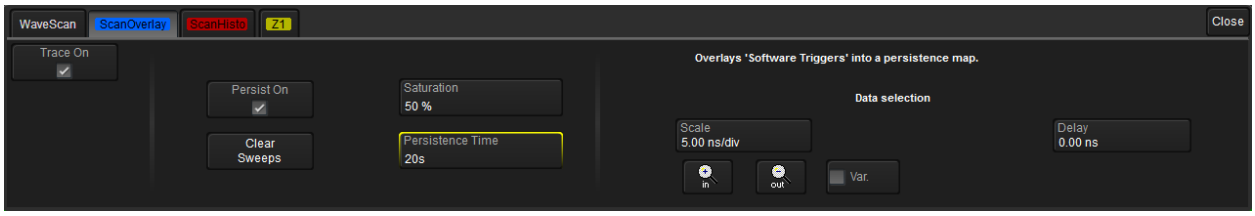
Scan Overlay

Scan Overlay view plots the location of captured events in a new trace.

To apply monochromatic persistence to the scan overlay:

1. Check **Persistence On**.
2. Enter a **Saturation** value. This controls...
3. Choose a **Persistence Time**. The higher the time, the more static the persistence display.

To rescale 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, 10.



Scan Histogram provides a statistical view of edges that meet your search criteria.

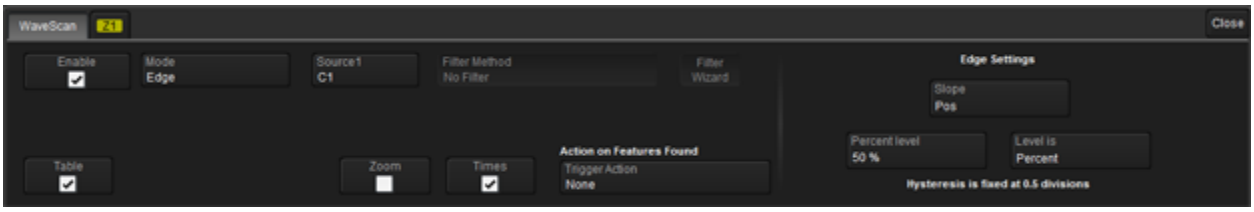
Zoom

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

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

Set 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.

History Mode

History Mode allows you to review any acquisition saved in the oscilloscope's history buffer, which automatically stores all acquisition records until full. Not only can individual acquisitions be restored to the grid, you can "scroll" backward and forward through the history at varying speeds to capture individual details or changes in the waveforms over time.

Each record is indexed and time-stamped, and you can choose to view the absolute time of acquisition or the time relative to when you entered History Mode. In the latter case, the last acquisition is time zero, and all others are stamped with a negative time. The maximum number of records stored depends on your acquisition settings and the size of the oscilloscope memory.

To view history:

1. Press the Front Panel **History Mode button**, or choose **Timebase > History Mode**.
2. Select **View History** to enable the history display, and **View Table** to display the index of records. Optionally, select to view **Relative Times** on the table.

3. Choose a single acquisition to view by entering its **Index** number on the dialog or selecting it from the table of acquisitions. You can also use the Navigation buttons or the slider bar at the bottom of the dialog to "scroll" the history of acquisitions.
    - The top row of buttons scrolls continuously and are (left to right): Fast Backward, Slow Backward, Pause, Slow Forward, Fast Forward.
    - The bottom row of buttons steps one record at a time and are (left to right): Back to Start, Back One, Go to Index (#), Forward One, Forward to End.

Entering History Mode automatically stops new acquisitions. To leave History Mode, press the Front Panel History Mode button again or clear the View History checkbox on the History dialog. Restart acquisition by pressing one of the Front Panel Trigger Mode buttons.

Pass/Fail Testing

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

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

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

Access Pass/Fail Test Dialogs

1. Choose **Analysis → Pass/Fail** to display the **Pass/Fail** dialog.
2. Touch the **Q*x* button** or tab where you want to set up the mask.

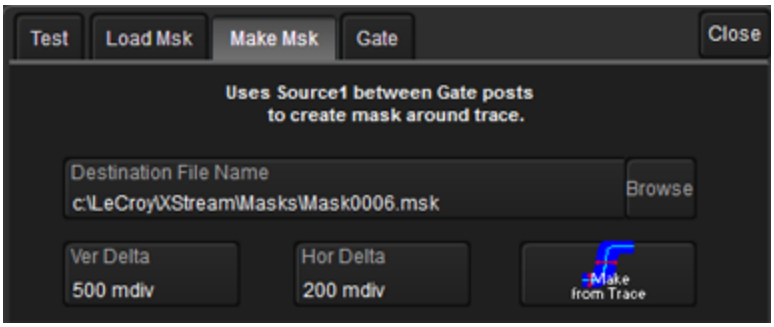
   The source waveform will be tested against this mask whenever Q*x* is enabled on the Pass/Fail dialog.
3. From the pop-up menu, select **Pass/Fail Condition** and **Mask test.**.

   The Qx 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 source waveform. The mask will cover the area of the waveform, plus the boundary values you enter.

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



2. If desired, enter a new **Destination File Name** and path, or touch Browse and select a previous file to overwrite. The file name should end with the **.msk** extension.
3. Touch the **Ver Delta** and **Hor Delta** fields and enter boundary values using the pop-up numeric keypad or the Front Panel Adjust knob.
4. Touch **Make from Trace**.

Load Mask

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

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

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

   OR

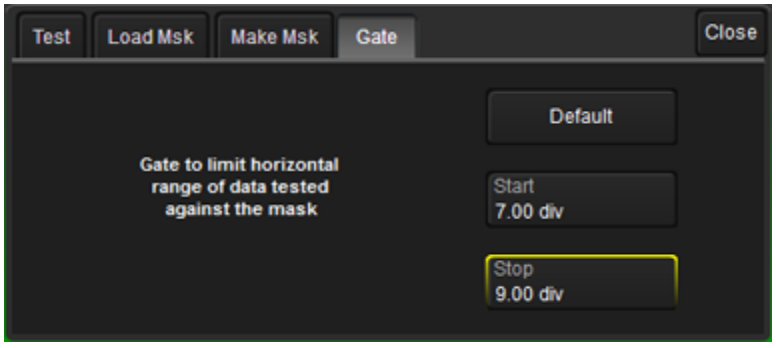
   To use an industry standard mask, touch **Standard** and select the mask from the pop-up.

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

Set Gates

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

1. Touch the **Gates tab** to display the dialog.

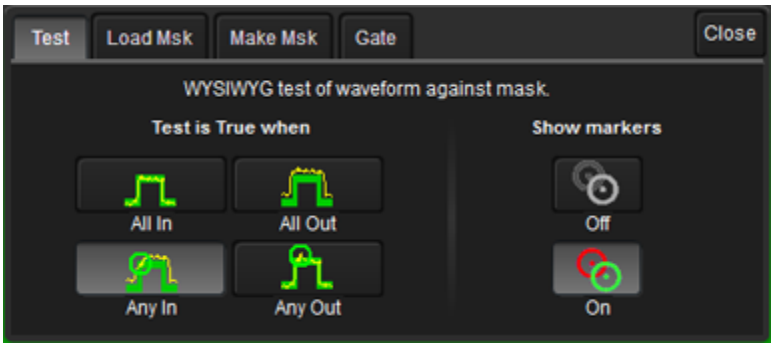


2. Enter the **Start** and **Stop** timebase divisions that mark the segment of the waveform to be tested with this mask. This can be a whole division of a fraction of a division. Divisions are numbered 1-*n* left to right.

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

Define Test

1. Touch the **Test tab** to display the dialog.



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

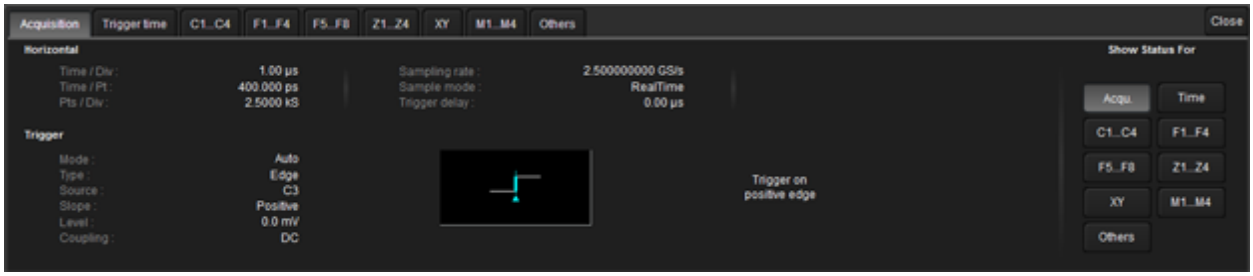
Removing a Mask from the Display

1. Access the**Qx** dialog where the mask is set.
2. On the **Load Mask** right-hand dialog, click the **Delete** button.

View Configurations

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

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



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 oscilloscope application.

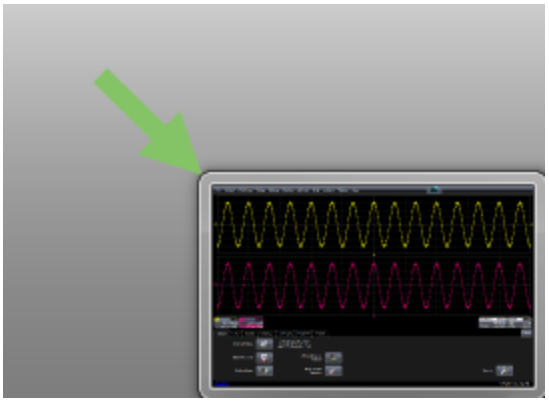
To access the Utilities dialog, choose **Utilities → Utilities Setup...** from the menu bar.

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

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

NOTE: Hardcopy Setup controls the behavior of the oscilloscope's 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 oscilloscope application window. Maximize the application by touching the oscilloscope display icon located at the lower-right of the desktop.

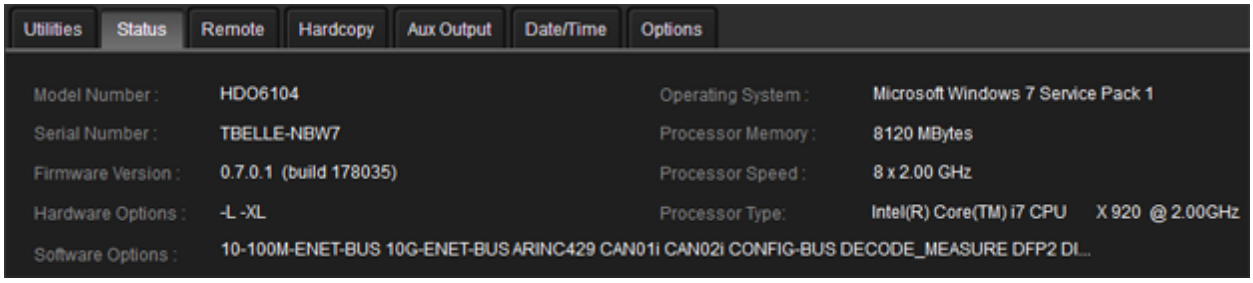


Touch-Screen Calibration launches a sequence of display calibration screens. You will be prompted through a series of actions to improve the precision and accuracy of the touch screen.

The **Service** button to the far right of the dialog (not shown) launches a section of the application reserved for qualified Teledyne LeCroy 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.



To view status, 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. That feature displays the current state of the oscilloscope configurations that affect the analysis functions--such as acquisition, channel, parameter, function, and memory settings.

Remote Control Settings

The Utilities Remote dialog contains settings to configure remote control of the instrument. Supported communication protocols are:

- **TCPIP (Ethernet)** - Enables remote control across a LAN via an Ethernet port. If you choose this option, you also need to install Teledyne LeCroy's VICP drivers on the controller. These are included in the VICP Passport plug-in, available free from teledynelecroy.com.

  NOTE: The instrument uses Dynamic Host Configuration Protocol (DHCP) as its addressing protocol. Therefore, it is not necessary to set up an IP address if your network supports DHCP. If it does not, you can assign a static address in the standard Windows network setup menu on the oscilloscope.

- **LXI (Ethernet)** - Enables remote control across a LAN via an Ethernet port.
- **USBTMC** - Enables remote control via a device connected to the USBTMC port.
- **GPIB** - Enables remote control using GPIB if you have the GPIB-USB adapter connected to any host USB port.

Set Up Remote Control

Contact your Network Administrator to connect the oscilloscope to your LAN. Use a USB cable to connect the oscilloscope directly to a PC.

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 you are using TCPIP or LXI, touch the **Net Connections** button. The Windows Network Connections window opens for you to select a network.
4. If you are using TCPIP and wish to restrict control of the oscilloscope to specific network clients, touch **Yes**. Enter the IP addresses or DNS names of the authorized controllers in a comma-delimited list.

Configure the Remote Control Assistant Event Log

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

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

Export Contents of the Event Log

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

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

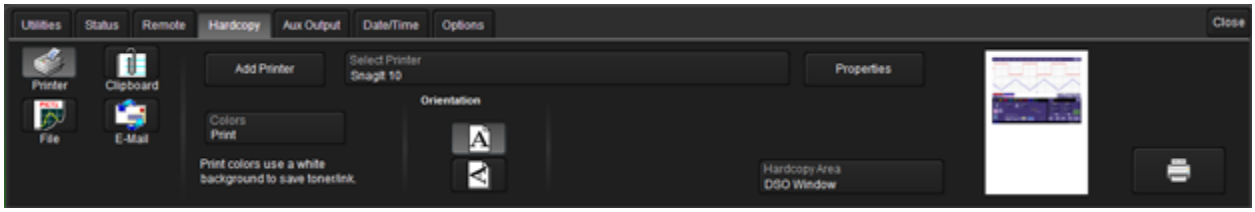
Print (Hardcopy) Settings

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

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

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

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



Send to Printer

Follow these procedures to configure output to a printer.

Add Printer

NOTE: Any printer compatible with Microsoft Windows Embedded Standard 7P operating system is supported by your instrument. Install printer drivers on the instrument outside of the oscilloscope application. Printers can be connected via LAN (Ethernet) or USB.

1. From the menu bar, choose **File → Print Setup...** or **Utilities → Utilities Setup → HardCopy**.
2. On the Utilities **Hardcopy** dialog, choose **Printer**.
3. Touch the **Add Printer** button that appears.

   A Microsoft Windows Devices and Printers window opens where you can configure a new printer.
4. To make the printer the instrument default, select it from the **Select Printer** list.

Print Setup

1. From the menu bar, choose **File → Print Setup...** or **Utilities → Utilities Setup → HardCopy**.
2. On the Hardcopy dialog, choose **Printer**.
3. 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.
4. Choose a page **Orientation**: portrait or landscape.
5. Optionally, choose a color scheme and hardcopy (print) area.
6. Optionally, touch **Properties** to open the Windows print dialog and adjust printer properties.

Print to File

Follow this procedure to print screen captures to a file. The default save directory can be set to any folder on the oscilloscope hard drive, or an external drive connected via USB port.

1. Choose **Utilities → Utilities Setup...** from the menu bar, then touch the **Hardcopy** tab.
2. On the Hardcopy dialog, choose **File**.
3. Choose the output **File Format**.
4. Enter a **File Name**. This will form the basis of all print filenames, until you change it.

   NOTE: Numbers at the end of the filename will be truncated, as the instrument appends numbers to this name with each new file. If you wish to add your own identifying numbers, place them at the front of the name.
5. Optionally, enter the path to a new save **Directory**, or touch the **Browse** button and navigate to the folder.

   NOTE: The default print folder is C:\...\XStream\Hardcopy. Other types of files that may be saved using other oscilloscope functions, such as masks and scripts, have their own XStream subfolders.
6. Optionally, choose a color scheme and hardcopy (print) area.

Copy to Clipboard

Follow this procedure to copy screen captures to the clipboard so you can paste them into another application (like Microsoft Word, for example).

1. Choose **Utilities → Utilities Setup...**, then touch the **Hardcopy** tab.
2. On the Hardcopy dialog, choose **Clipboard**.
3. 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. Choose **Utilities → Utilities Setup...** from the menu bar, then touch **Hardcopy** tab.
2. On the Hardcopy dialog, choose **E-Mail**.
3. Choose the output **File Format**.
4. If you wish to be able to include messages with the files as they are sent, check **Prompt for message to send with mail**.
5. Optionally, choose a color scheme and hardcopy (print) area.
6. 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

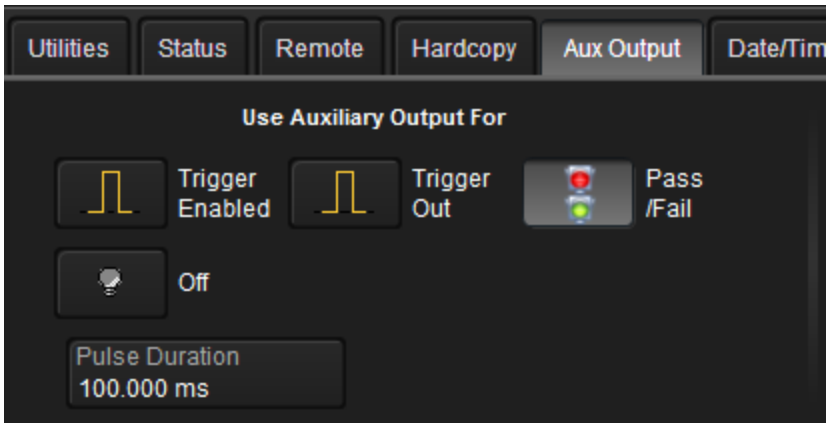
To limit which part of the touch screen is captured, touch **Hardcopy Area** on the Hardcopy dialog and choose from:

- **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 and Cal Out ports.

Configure Auxiliary Output



Choose one of the following under **Use Auxiliary Output For**:

Trigger Enabled - can be used as a gating function to trigger another instrument when the oscilloscope is ready.

Trigger Out - can be used to trigger an external oscilloscope off the instrument's state.

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

Off - disables auxiliary input/output.

Configure Calibration Output

A calibration signal can be output from the Cal Out hook on the front of the oscilloscope. Choose one of the following under **Use Calibration Output For**:

Square - sends a square wave signal. With this selection, also enter the wave **Frequency** and **Amplitude into 1 MΩ**. The **Set to 1 kHz, 1 V Square Wave** button does exactly that.

DC Level - sends a reference level. Enter an **Amplitude into 1 MΩ**.

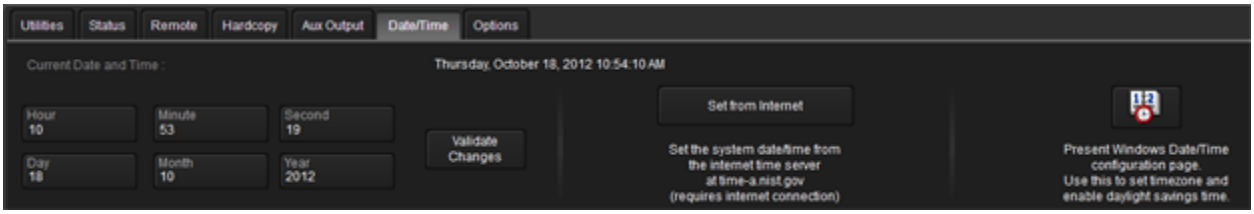
Off - disables calibration output.

Date/Time Settings

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

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

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



Manual Method

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

Internet Method

This method uses the Simple Network Time Protocol (SNTP) to read the time from time-a.nist.gov. The oscilloscope must be connected to an internet access device through the LAN (Ethernet) port on your instrument's I/O panel. .

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

Windows Method

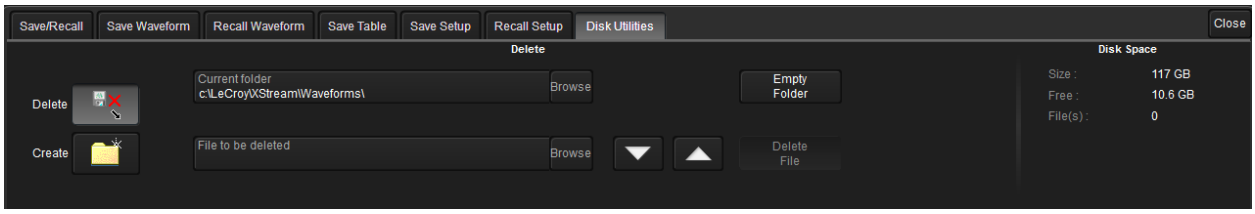
To set date and time using the internal Windows system clock, touch the **Windows Date/Time** button. This displays the standard Windows **DateTime Properties** pop-up dialog, where you can further configure these settings. If you are satisfied with the setup, just touch OK.

Disk Utilities

Use the Disk Utilities dialog to arrange the file/folder structure on your instrument's hard drive. You can delete files, folders, or create new folders.

NOTE: All Disk Utilities can also be accomplished using the standard Microsoft Windows file management tools.

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



Delete a Single File

1. Touch the **Delete** button.
2. Touch **Current folder** and provide the path to the folder containing the file, or touch **Browse** and navigate to the folder.
3. Touch **File to be deleted** and provide a file name.

   NOTE: You can also use the **up down arrows** to move through the files contained in the **Current folder** selected.
4. With the desired file selected, touch **Delete File**.

Delete All Files in a Folder

1. Touch the **Delete** button.
2. Touch **Current folder** and provide the path to the folder, or touch **Browse** and navigate to the folder.
3. With the desired folder selected, touch **Empty Folder**.

Create a New Folder

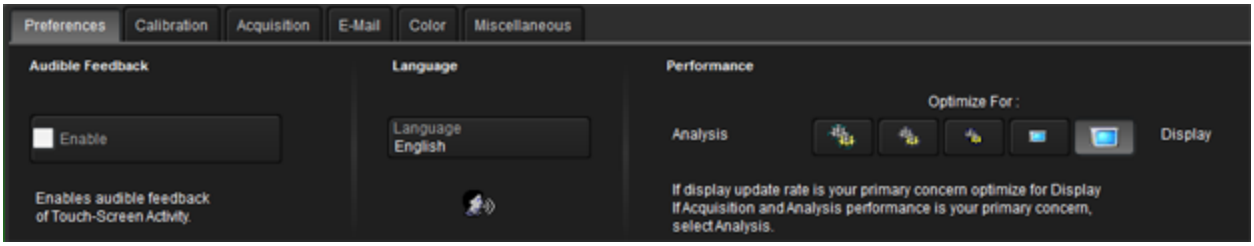
1. Touch **Create**.
2. Touch **Current folder** and provide the full path to the new folder, including the folder name.
3. Touch **Create Folder**.

NOTE: Disk Space data for Size, Free, and File(s) is available on the far right of the dialog for convenience.

Preferences Settings

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

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



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

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

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

Language sets the language used on the display.

Performance settings let you optimize oscilloscope 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 Calibration, Acquisition, E-Mail, Color, and Miscellaneous settings.

Calibration Settings

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

Manually calibrate the oscilloscope when:

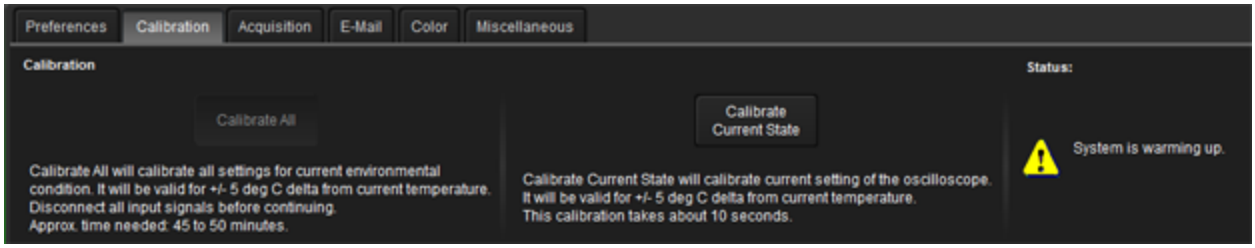
- It is used in temperatures that differ from the previous calibration temperature by more than 5° C.

  OR

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

Go to **Utilities > Calibration Setup**.

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



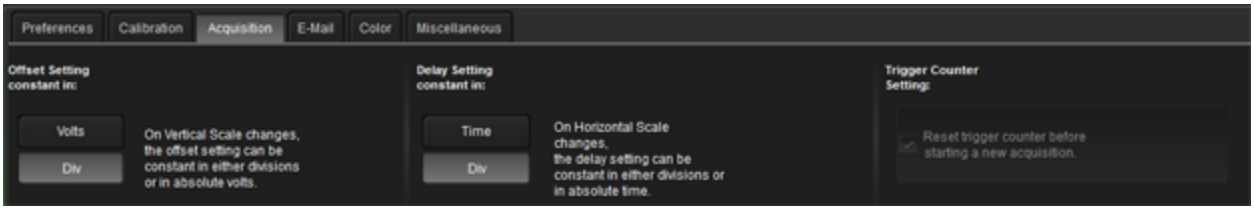
There are two options for this calibration: Calibrate All or Calibrate Current Setting.

- **Calibrate All**—All possible combinations of vertical and horizontal settings are calibrated at the current temperature. This calibration is valid for the current temperature ± 5° C and takes about 50 minutes.
- **Calibrate Current Setting**—The oscilloscope is calibrated at the current vertical and horizontal setting. This calibration is valid for this setting for the current temperature ± 5° C and takes under 30 seconds.

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

Acquisition Settings

The Preferences Acquisition settings determine how traces behave on screen as gain or timebase changes. Make a selection in each area.



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.

Trigger Counter Setting:

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

Color Settings

Preferences Color 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.

E-Mail

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

E-Mail Server- Choose a server protocol from the following options:

- **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.

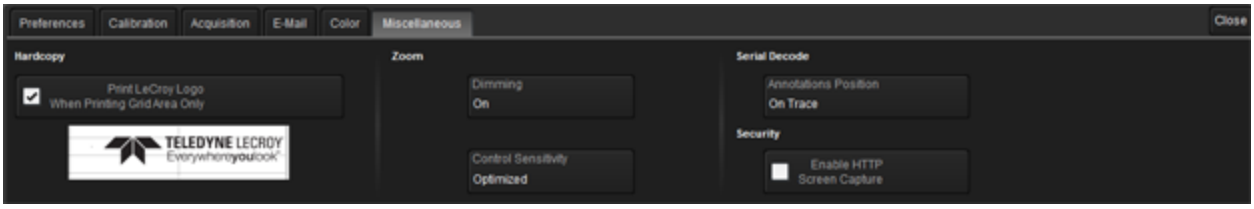
Configuration - Provide values based on your selected server protocol:

- **If you chose MAPI**, touch inside the **Originator Address (From:)** data entry field and provide the instrument's e-mail address. Then touch inside the **Default Recipient Address (To:)** data entry field and provide the recipient's e-mail address.
- **If you chose SMTP**, touch inside the **SMTP Server** data entry field and provide the name of your server. Touch inside the **Originator Address (From:)** data entry field and provide the instrument's e-mail address. Then touch inside the **Default Recipient Address (To:)** data entry field and provide the recipient's e-mail address.

Send Test Mail - Click this button to send a confirmation message to ensure proper e-mail configuration.

Miscellaneous Settings

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



Hardcopy: You can add the Teledyne LeCroy logo to print output by checking **Print LeCroy Logo When Printing Grid Area Only**. This identifies the instrument as the source of the image.

Zoom: You can adjust zoom behavior as follows:

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

Serial Decode Annotation Position: If you have Serial Trigger or Decode options installed on your oscilloscope, this control determines the placement of annotation labels relative to the trace line.

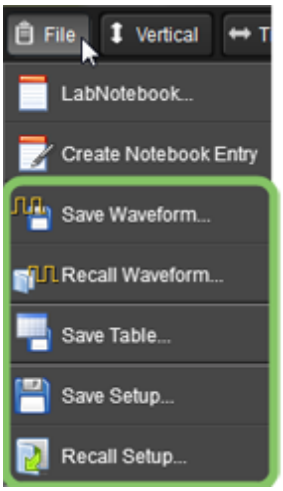
- **On Trace** places the label close to the line.
- **On Noisy Trace** sets the label further from the line to accommodate potential noise spikes in the trace.

Security: Check **Enable HTTP Screen Capture** to enable remote capture of the oscilloscope display over the Internet.

Save/Recall Overview

The Save/Recall features allow for storage and retrieval of Waveforms, Table Data, and Instrument Setups. There are also Disk Utilities for arranging the file/folder structure on your instrument's hard drive.

Directly access the **Save Waveform**, **Save Table**, **Save Setups**, and **Disk Utilities** dialogs from the **File** menu.



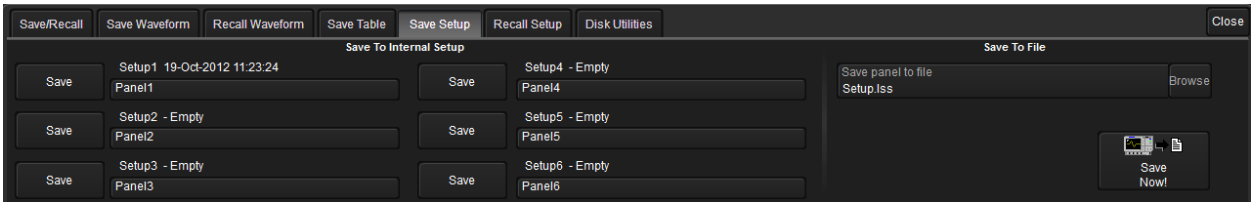
A main Save/Recall dialog is shown providing buttons to access the aforementioned dialogs for specific Save/Recall functions.

Save/Recall Setups

The Save/Recall Setup dialogs allow for quick saving and recalling of up to six oscilloscope panel settings internally on your instrument. If desired, you can also save and recall your oscilloscope panel settings as an .lss file to a specific hard disk location, a network location, or USB drive.

Saving Oscilloscope Setups

Access the **Save Setup...** dialog by either selecting **File → Save Setup...** from the menu bar or clicking the **Save Instrument Setup** button on the main Save/Recall dialog.



SAVE SETUP TO MEMORY

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

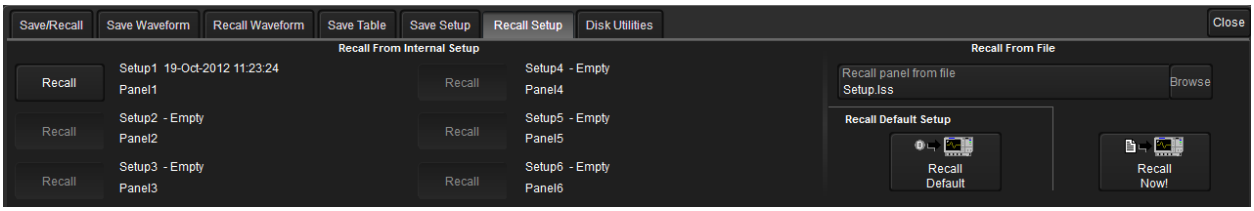
Settings are saved to the D:\Internal Setups location on your instrument. The save date/time is displayed above the **SetupX** data entry control.

SAVE SETUP TO FILE

1. Touch **Save Panel to File** and enter the full path to the file, or touch **Browse** to navigate to the file folder.
2. Enter a filename, or choose a existing file to overwrite.
3. Touch **Save Now!**

Recalling Oscilloscope Setups

Access the **Recall Setup...** dialog by either selecting **File → Recall Setup...** from the menu bar or clicking 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 name and save date/time. Otherwise, the slot is labeled **Empty**.

RECALL SETUP FROM FILE

1. Touch **Recall panel from file** and provide the path to the destination file, or touch **Browse** to navigate to the file.
2. Touch **Recall Now!**

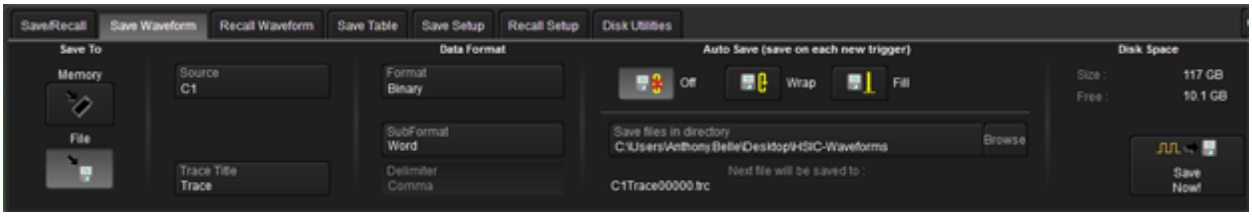
Save/Recall Waveforms

Save Waveform

The Save Waveform dialog is used to save traces to either an internal memory location, or to a text or binary format file. The source waveform can be any trace; for example, a channel, math function, or a waveform stored in memory.

Access the **Save Waveform** dialog by either selecting**File → Save Waveform** from the menu bar or clicking the **Save Waveform** button on the main Save/Recall dialog.

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



Save Waveform To Memory

1. Touch **Memory**.

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

2. Choose the **Source** trace you are saving.
3. Choose the **Destination** location.
4. Optionally, touch **Trace Title** to change the default name of your waveforms.

   CAUTION. Any numbers placed at the end of this name are truncated because the instrument automatically appends the next number in sequence to each file. if you want to use your own identifying number, place it at the beginning of the name, or append an alpha character at the end of the number: XYZ32a, for example.

5. Touch **Save Now!**

Save Waveform To File

1. Touch **File.**
2. Choose the **Source** waveform.
3. Touch **Data Format** and select a file format.
   - **Binary** - saves the file to Teledyne LeCroy's binary file format. This format is documented in various Remote Control manuals for Teledyne LeCroy oscilloscopes. Binary results in the smallest possible file size, and is recommended when 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 output file (.txt extension).
   - **MATLAB** - Text output file compatible with MATLAB (.dat extension).
   - **Excel** - Text output file compatible with Excel (.csv extension).
   - **MathCad** - Text output file compatible with Excel (.prn extension).
   - **Audio** - .wav file.
   - **WaveML** - .xml file, used for persistence traces.
4. Depending on your file format selection, you may also need to specify a **SubFormat**.
   - **Word** - For Binary files, specifies the samples in the output file are represented with 16 bits. Always use this option unless Byte mode is "pre".
   - **Byte** - For Binary files, specifies the samples in the output file are represented with 8 bits. This option can result in a loss of output file resolution.
   - **Auto** - For Binary files, looks at the data and automatically selects either Word or Byte subformat.
   - **Amplitude only** - Specifies the output file include amplitude data for each sample, but not sample time information.

- **Time and Amplitude** - Specifies the output file include both time and amplitude data for each sample.
- **With Header** - Specifies to include a header with scaling information.

5. If you selected **ASCII** file format, also touch the **Delimiter** control and select a delimiter character from the pop-up menu. Choose from **comma**, **space**, **semicolon**, or **tab** delimiters.
6. Touch **Save Now!**

AUTOSAVE

The **Auto Save** feature automatically saves a waveform to disk after each new trigger.

Enable Auto Save from the **Save Waveform** dialog by selecting one of the **Auto Save** buttons: **Wrap** (old files overwritten) or **Fill** (no files overwritten).

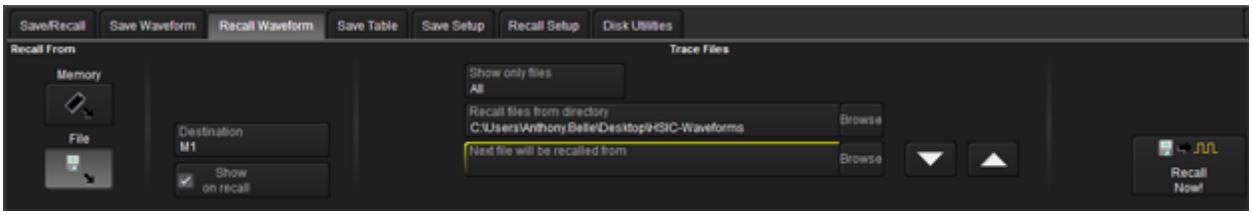
Choosing **Off** diables AutoSave.

Touch the **Browse** button next to the **Save file in directory** control and navigate to the location where you want the file saved. The file name is assigned automatically and is shown under the control.

Recall Waveform

Access the **Recall Waveform** dialog by either choosing **File → Recall Waveform** from the menu bar or clicking the **Recall Waveform** button on the main Save/Recall dialog.

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



RECALL WAVEFORM FROM MEMORY

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

RECALL WAVEFORM FROM FILE

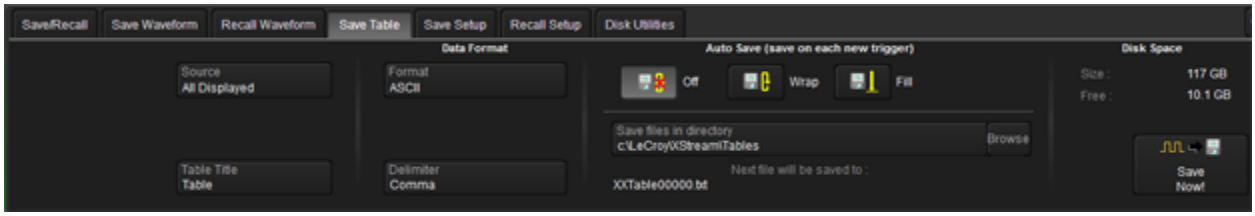
1. Select **File**.
2. Touch **Recall files from directory** and enter the path to the waveform folder, or touch **Browse** and navigate to the folder.
3. Use the **Up /Down Arrows** to cycle through the available files until the desired file is selected.

   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.

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 **Trace Title** and enter a new base filename. This name with a sequence number appended to it will be used for all filenames until you change it.

   CAUTION. Any numbers placed at the end of this name are truncated because the instrument automatically appends the next number in sequence to each file. if you want to use your own identifying number, place it at the beginning of the name, or append an alpha character at the end of the number: XYZ32a, for example.

3. Touch **Data Format** and choose from:

   ASCII -.txt extension.

   Excel - .csv extension.

4. If you selected **ASCII** format, also touch **Delimiter** and choose a character.
5. Optionally, to automatically save data to this file type after each new trigger, choose an **Auto Save**option: **Wrap** overwrites old files in the order created; **Fill** stops before overwriting files.

   CAUTION. Because the hard disk is partitioned, if you have frequent triggers, it is possible you will eventually run out of storage space on the D: drive. Choose Wrap only if you're not concerned about files persisting on the instrument. If you choose Fill, plan to periodically move files out of the waveform directory.

6. Optionally, to change the save directory, touch the **Browse** button and navigate to the folder.
7. Touch **Save Now!**

LabNotebook

Teledyne LeCroy's LabNotebook feature extends the documentation capabilities of your oscilloscope. It allows you to create and save Notebook Entries containing all displayed waveforms, the oscilloscope setup under which they were taken, and custom annotations. Notebook Entries can then be output to a hardcopy report format—.pdf, .rtf, or .html—and printed or e-mailed. You can also configure your own report layout if you prefer not to use the default, and even substitute your own company logo in the header.

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

The Flashback Recall feature instantly recalls the setups stored with individual Notebook Entries, enabling you to restore the exact state of the oscilloscope at a later date to perform additional analysis. A keyword filter makes it easy to find and recall a specific Notebook Entry.

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

Create Notebook Entry

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

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. In that case, you can select the entry from the list of Notebook Entries to manage them at a later time.

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. Choose **File > Create Notebook Entry**.

   OR

   Press the **Front Panel Print button** if you have configured it for LabNotebook.
4. Optionally, **Enter Report Title** and **Description**.

   The default title is the date and time stamp. You can leave this as is, append some descriptive text to it, or completely remove it from your title.
5. Touch **Save**.
6. Use the Drawing toolbar to markup the Notebook Entry. Click **Done** when finished.

LabNotebook Drawing Toolbar

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

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

From left to right, the tools are:

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

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

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

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

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

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

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

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

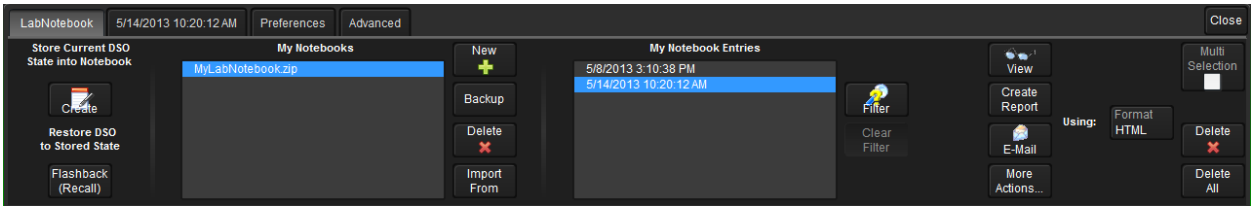
Move Toolbar Button undocks the drawing toolbar so you can move it anywhere on the display. This helps to keep tools handy when working on a particular area of a waveform. Touch the button again to restore the toolbar to the top of the Drawing window.

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

Manage Notebook Entries

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

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



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

Select Notebook Entries

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

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

   OR

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

Filter Notebook Entries

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

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

View Notebook Entries

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

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

Edit Notebook Entries

1. Select the notebook from the **My Notebooks** list.
2. Select the entry from the **My Notebook Entries** list.
3. Go to the **second tab** labeled with the entry name.
4. Modify the **Title** or **Description**.
5. To add markup to the entry, touch the **Scribble** button and use the Drawing Toolbar.

Email Notebook Entries

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

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

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

Print Notebook Entries

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

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

Delete Notebook Entries

Use the **Delete** button to remove selected Notebook Entries, or **Delete All** to clear the entire **My Notebook Entries**. Be aware that unless you have previously backed up the notebook, the 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. Now select it.
6. Enter a **File name** for the report.
7. Click **OK** to create the report.

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 when it is selected. 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 from that list.

Back Up Notebook

1. Choose **File > LabNotebook**.
2. Select the notebook from the My Notebooks list.
3. Touch the **Backup** button next to the My Notebooks list.
4. Optionally, enter a new **File Name** or choose a new storage **Folder**.

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

Import Notebook

Archived notebooks can be imported into the working 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.

NOTE: The File menu Print option will continue to use whatever method you have set on the Utillities Hardcopy tab when invoked. Go to Utilities > Utilities Preferences > Hardcopy to make that selection.

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

Flashback Recall

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

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

Some result data not included in Flashback Recall are:

- **Persistence data** - While it is saved in hardcopy and is printed on the report, it is not recalled during Flashback.
- **Histogram data** - 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** - Certain math operations result in the creation of floating point waveforms with much higher resolution than can be stored in a 16-bit waveform file. This extra resolution is not preserved when traces are recalled using flashback.
- **Cumulative Measurements** - Any measurements on when the Lab Notebook entry is created are not saved individually in the database (other than being embedded in the hardcopy image). This means that when flashback is used, the measurements are recomputed using the recalled waveform data. Normally, doesn't pose a problem; however, if cumulative measurements were on when the entry was stored and the cumulative measurements accumulated data from multiple acquired waveforms, they lose their history and show instead only the results from the stored waveforms.

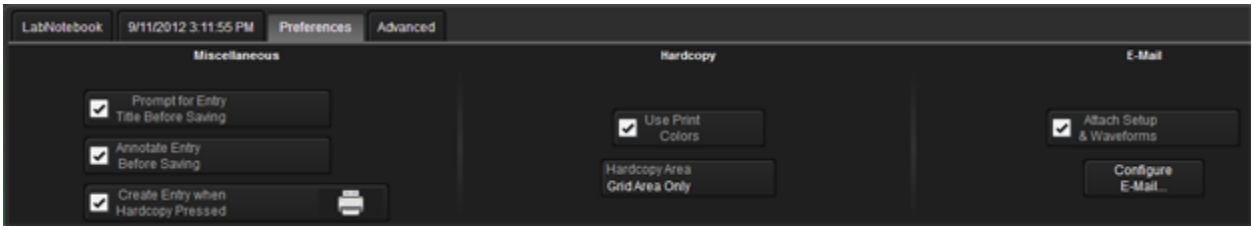
Customize Report

Teledyne LeCroy provides a default LabNotebook report template which includes 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 C:LeCroy\XStream\Xport folder.
2. Choose **File > LabNotebook**, then touch the **Advanced** tab.
3. Deselect the **Use Default** checkbox next to the Logo field.
4. Touch the **Browse** button and navigate to the the new logo file. Select and touch **OK**.The new file path appears in the Logo field with a preview of the image above it.

Configure LabNotebook Preferences

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



Select/deselect the following options:

Prompt for Report 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 Report Before Saving opens the Drawing Toolbar to annotate a notebook entry as soon as it is created.

Save Report 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 reports.

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

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

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

Maintenance

Cleaning

Clean only the exterior of the oscilloscope and probes using a soft cloth moistened with water. If required, use an aqueous solution of 75% isopropyl alcohol for more efficient cleaning. Exercise care to avoid scratching the display. Unplug the power cord from the AC outlet before cleaning.

CAUTION. Do not attempt to clean internal parts. Avoid getting moisture inside the unit during external cleaning. Do not use any abrasive or chemical cleaning agents.

Calibration

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

See Calibration Settings for an explanation of the manual calibration options.

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

Touch Screen Calibration

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

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

Language Selection

To change the language that appears on the display, go to **Utilities > Preference Setup > Preferences** and make your Language selection. Reboot the oscilloscope after changing the language.

To also change the language of the Windows operating system:

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

   NOTE: Other language packs are available from Microsoft's website.
7. Touch the **oscilloscope icon** in the lower-right corner of the desktop to restore the display.

Add Software Option

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

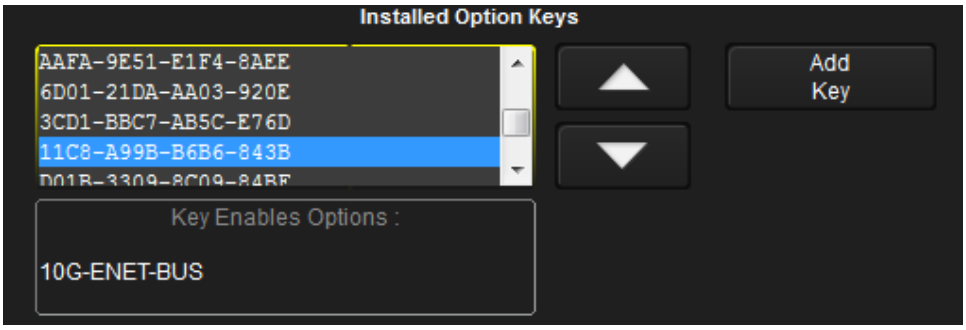
Contact your local Teledyne LeCroy representative or national distributor to purchase an option. You will receive a Key Code by email that enables the new functionality.

To install the key and activate the software:

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

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

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



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

X-Stream Firmware Update

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

The firmware update procedure *does not* modify or delete any saved panel setups, waveforms, screen captures, calibration constants, or other data stored on the oscilloscope's D: drive.

Downloading and Installing the Firmware

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

   Click **Install** when done.

   NOTE: Not all components are available on all models. If you do not see an option listed, it is not required for your oscilloscope.
9. If you receive Windows security warnings, **trust** and **Install** the file. If you see the Hardware Programmers screen, accept all code installations, then click **Close** to return to the X-Stream Setup wizard.
10. When installation is complete, choose **Reboot now** and click **Finish**.

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

HDO System Recovery

Using Acronis True Image Home

Your HDO oscilloscope is designed to operate reliably for many years. The application software operating the instrument runs on a Microsoft Windows® 7 Professional for Embedded Systems platform. The loading or incomplete removal of additional Windows applications can cause problems in the stability of the operating system. Severe cases may require you to reload the base operating system and oscilloscope application.

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

The following procedures cover the basic steps for restoring the oscilloscope using Acronis True Image Home ver. 2011. **Follow each procedure completely in the order shown.** Since third-party recovery software is used, our instructions may not be the most definitive or current. We encourage you to take advantage of additional resources available from the Acronis website:

Acronis True Image Home
www.acronis.com/homecomputing/download/docs/

Before You Start

- Connect a keyboard and mouse via the USB host ports.

Run the Recovery Wizard

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

   NOTE: Do not press F11 before you see the "Starting Acronis Loader..." message or you will enter the boot device selection menu. If a bootmenu dialog box appears, press Cancel or Esc.
3. Select **Acronis True Image Home (Full Version)**.
4. On the Acronis True Image Home page, under options for Recover, select **My Disks**. The Recovery Wizard opens.
5. On the Recovery Wizard, under Archive Selection, select the disk archive that has a create date, then click **Next**.
6. Under Recovery Method, select **Recover whole disks and partitions**, then click **Next**.
7. Under What to Recover, select **NTFS (SYSTEM) (C:)**, then click **Next**.
8. Under Settings of Partition C, in the top section, Partition location (required), select **New Location**.The Partition Destination window opens.
9. Under New Partition Location, select **NTFS (SYSTEM) (D:)**, then click **Accept**. This returns you to the Settings of Partition C step. Click **Next**.
10. A summary window is displayed indicating that Acronis True Image is ready to proceed with Recovering partition C -> D. Click **Proceed** to start the recovery process.

NOTE: Recovery takes approximately 4 to 15 minutes depending on the version of operating system that is being restored. The progress is displayed on the screen.

11. When recovery is complete, you will see the message "Recover operation succeeded." Click **OK**.
12. Click the **close button** to exit the Acronis window. The oscilloscope will restart and begin installing the required software.

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

Reinstall the Oscilloscope Application

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

NOTE: You may see a Windows Security message indicating that Windows can't verify the publisher of the driver software. Choose "Install this driver software anyway" and click Install.

4. When the X-Stream installation is complete, choose to **Reboot now** and click **Finish**.

Reactivate the F11 Startup Utility

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

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

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

Restore Software to Current Revision Levels

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

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

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

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

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

Technical Support

Phone

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

Web

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

teledynelecroy.com/support/techhelp.

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 download Oscilloscope System Recovery Tools and Procedures, which contains instructions for using Acronis® True Image® Home included with the oscilloscope.

Returning a Product for Service

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

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

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

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

Contact Teledyne LeCroy

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

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

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

Singapore, Oscilloscopes
Teledyne LeCroy Singapore Pte Ltd.
Blk 750C Chai Chee Road #02-08
Technopark @ Chai Chee
Singapore 469003
Ph: ++ 65 64424880
FAX: ++ 65 64427811

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

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

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

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

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

Certifications

This section certifies the instrument's Electromagnetic Compatibility (EMC), Safety and Environmental compliances.

EMC Compliance

EC Declaration of Conformity- EMC

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

EN 61326-1:2006, EN 61326-2-1:2006 EMC requirements for electrical equipment for measurement, control, and laboratory use. [1]

ELECTROMAGNETIC EMISSIONS:

CISPR 11:2003, Radiated and Conducted Emissions Group 1, Class A [2 3]

EN 61000-3-2:2006 Harmonic Current Emissions, Class A

EN 61000-3-3/A2:2005 Voltage Fluctuations and Flickers, Pst = 1

ELECTROMAGNETIC IMMUNITY:

EN 61000-4-2:2001 Electrostatic Discharge, 4 kV contact, 8 kV air, 4 kV vertical/horizontal coupling planes [4]

EN 61000-4-3:2006 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 [4]

EN 61000-4-4:2004 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 61000-4-6:2007 RF Conducted Electromagnetic Field, 3 Vrms, 0.15 MHz - 80 MHz [4]

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

1 To ensure compliance with all applicable EMC standards, high quality shielded interface cables should be used.
2 Emissions which exceed the levels required by this standard may occur when the oscilloscope is connected to a test object.
3 This product is intended for use in nonresidential areas only. Use in residential areas may cause electromagnetic interference.
4 Meets Performance Criteria “B” limits of the respective standard: during the disturbance, product undergoes a temporary degradation or loss of function or performance which is self-recoverable.
5 Performance Criteria “C” applied for 70%/25 cycle voltage dips and for 0%/250 cycle voltage interruption test levels per EN61000-4-11.

EUROPEAN CONTACT:

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

Australia & New Zealand Declaration of Conformity– EMC

Oscilloscope complies with the EMC provision of the Radio Communications Act per the following standards, in accordance with requirements imposed by Australian Communication and Media Authority (ACMA):

CISPR 11:2003 Radiated and Conducted Emissions, Group 1, Class A, in accordance with EN61326-1:2006 and EN61326-2-1:2006.

AUSTRALIA / NEW ZEALAND CONTACTS:

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

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

Safety Compliance

EC Declaration of Conformity– Low Voltage

The oscilloscope meets intent of EC Directive 2006/95/EC for Product Safety. Compliance was demonstrated to the following specifications as listed in the Official Journal of the European Communities:

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

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

The design of the instrument has been verified to conform to the following limits put forth by these standards:

- Overvoltage Category II: equipment intended to be supplied from the building wiring with a nominal supply voltage up to 300V.
- Measurement Category 0: oscilloscope measurement terminals that are not intended to be directly connected to the MAINS supply.
- Pollution Degree 2: operating environment where normally only dry, non-conductive pollution occurs. Occasionally a temporary conductivity that is caused by condensation must be expected. This location is a typical office/home environment.
- Protection Class I: grounded equipment in which protection against electric shock is achieved by basic insulation and a connection to the protective ground conductor in the building wiring.

U.S. Nationally Recognized Agency Certification

The oscilloscope has been certified by Underwriters Laboratories (UL) to conform to the following safety standard and bears UL Listing Mark:

UL 61010-1 Third Edition – Safety standard for electrical measuring and test equipment.

Canadian Certification

The oscilloscope has been certified by Underwriters Laboratories (UL) to conform to the following safety standard and bears cUL Listing Mark:

CAN/CSA-C22.2 No. 61010-1-12. Safety requirements for electrical equipment for measurement, control and laboratory use.

Environmental Compliance

End-of-Life Handling

The instrument is marked with this symbol to indicate that it complies with the applicable European Union requirements to Directives 2002/96/EC and 2006/66/EC on Waste Electrical and Electronic Equipment (WEEE) and Batteries.

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

Restriction of Hazardous Substances (RoHS)

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

ISO Certification

Manufactured under an ISO 9000 Registered Quality Management System. Visit teledynelecroy.com to view the certificate.

Warranty

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

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

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

Teledyne LeCroy shall not be responsible for any defect, damage, or failure caused by any of the following: a) attempted repairs or installations by personnel other than Teledyne LeCroy representatives or b) improper connection to incompatible equipment, or c) for any damage or malfunction caused by the use

of non-Teledyne LeCroy supplies. Furthermore, Teledyne LeCroy shall not be obligated to service a product that has been modified or integrated where the modification or integration increases the task duration or difficulty of servicing the oscilloscope. Spare and replacement parts, and repairs, all have a 90-day warranty.

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

Windows License Agreement

The HDO6000Oscilloscope software runs on a Windows® operating system. Teledyne LeCroy's agreement with Microsoft® prohibits users from running software that is not relevant to measuring, analyzing, or documenting waveforms on Teledyne LeCroy oscilloscopes.

700 Chestnut Ridge Road
Chestnut Ridge, NY 10977
USA

teledynelecroy.com