TELEDYNE LECROY
Everywhereyoulook™

Operator's Manual

WaveJet Touch Oscilloscopes

WaveJet Touch Oscilloscopes Operator's Manual

924497 Rev A
December, 2014

Contents

Figures

Tables

Welcome

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

The manual is arranged in the following manner:

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

Our website maintains the most current product specifications and should be checked for frequent updates.

Remember...

When your product is delivered, verify that all items on the packing list or invoice copy have been shipped to you. Contact your nearest Teledyne LeCroy customer service center or national distributor if anything is missing or damaged.

We can only be responsible for replacement if you contact us immediately.

Thank You

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

Sincerely,

David C. Graef

Teledyne LeCroy
Vice President and Chief Technology Officer

Safety Instructions

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

Symbols

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

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

High voltage **WARNING**. Risk of electric shock or burn.

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 (CH1-CH4 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

Specification Guarantee Limits:

Temperature: 10° to 35° C.

Humidity: ≤ 80% RH (no dew condensation allowed)

Altitude: Up to 3,048 m (10,000 ft)

Operational Guarantee Limits:

Temperature: 0° to 40° C.

Humidity: 80% RH at ≤ 30° C, decreasing to 55% RH at 40° C (no dew condensation allowed)

Altitude: Up to 3,048 m (10,000 ft)

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.

Power

AC Power

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

Power Consumption

Maximum power consumption is 50 W. Power consumption in standby mode is 2 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.

WaveJet Touch Overview

Front of the Oscilloscope

Figure 1: Front of oscilloscope

A. Touch Screen Display

The display shows the observed waveforms and related readouts, menus, trigger information, and status messages. The 7.5-inch VGA color LCD screen has resolution of 640×480 dots, with rich colors (16 or more) used in waveforms and channel information.

See Touch Screen for a full description of the screen elements.

B. Input/Output Panel

POWER switch turns on the power. When pressed (down), the oscilloscope is in ON mode. When in the “up” position, the oscilloscope is in Standby mode.

USB port can be used to send and receive data with external (USB) memory.

BNC input connectors are signal inputs to the oscilloscope. From left to right, they are CH1, CH2, CH3, and CH4. During XY waveform measurement, X is assigned to CH1 and Y to CH2.

EXT terminal is for input of external trigger and clock signals.

CAL terminal is for output of calibration signals. The upper part is for CAL signal output and the lower part is the GND. It is mainly used for probe calibration. The signal is a 3.0 Vp-p, 1 kHz square wave.

C. Control Panel

The control panel (also called the front panel) works in conjunction with the touch screen display It contains these major control groups:

TOP ROW

AUTO SETUP button automatically sets the vertical scale, timebase, and trigger conditions based on the input signals.

PRINT button captures the screen and handles it according to the Print menu setting. PRINT can be configured to automatically save the screen capture as a file on an external USB drive or print directly to a printer via the rear panel USB port.

UTILITIES button opens the Utilities menu, where you can set up printer options, remote control, and many other oscilloscope functions.

DISPLAY button opens the Display menu, where you can set up waveform type (VT or XY), vector (line style), intensity, persistence, and color gradient.

SAVE/RECALL button can save each channel's waveform or setup data to USB memory or internal memory (SAVE), or retrieves them (RECALL).

ADJUST

The **ADJUST knob** can be used to modify numeric values on touch menus that are marked with the gear icon  . Pressing the knob like a button toggles between FINE and COARSE increments. The ADJUST knob will have slightly different behavior in other cases; these are described later in this manual.

REPLAY

When acquisition is stopped the oscilloscope automatically enters Replay mode. Turn the **REPLAY knob** to go back or forward through the available sweeps, “scrolling” the acquisition.

Miscellaneous

CURSORS button turns on/off the cursor display and opens the Cursors setup menu.

MEASURE button opens the Measure setup menu.

REF button opens the Reference (Waveform) setup menu.

CLOSE MENU button returns to the previous touch menu. If there is no higher-level menu, it simply closes the menu.

HELP button launches a help window describing the operation of the next button or knob you use.

CLEAR button clears the following: sweeps, averaging, persistence, measurement statistics (Max, Min, Num), Pass/Fail result count, event log.

Trigger

The **LEVEL knob** controls the trigger level.

SETUP button opens the Trigger menu to set up trigger type, source channel, slope, coupling, and hold off.

AUTO, NORMAL, and SINGLE buttons set the respective trigger modes.

RUN/STOP button pauses or restarts acquisition.

Horizontal

Horizontal **DELAY knob** adjusts the trigger delay for all channels.

SETUP button opens the Horizontal setup menu.

Time/div knob (Horizontal Scale) selects the timebase range for all channels.

Vertical

Channel Number (CH) buttons turn on/off the waveform traces and open the Channel setup menu. Each button has a corresponding **Vertical OFFSET knob** and **Volts/div knob** that controls only that channel.

Math

The **MATH button** turns on/off a Math waveform and opens the Math setup menu.

The Math **OFFSET knob** controls the Math trace Vertical offset.

Math **Range Control knob** controls the scale for the Math trace.

Back of the Oscilloscope

Figure 2: Back of oscilloscope

A. **GPIB connector** can be used to connect to a computer for remote control.
B. **USB port** can be used to connect to a computer for remote control or to a printer.
C. **Ethernet port** (LAN interface) can be used to connect to a computer for remote control.
D. **Kensington lock** can be used to secure the instrument.
E. **Aux Out connector** can be configured for Trigger or Pass/Fail output.
F. **AC Power Inlet** connects the AC power cord.

Touch Screen

The WaveJet Touch employs a color LCD screen (VGA 640 × 480 resolution).

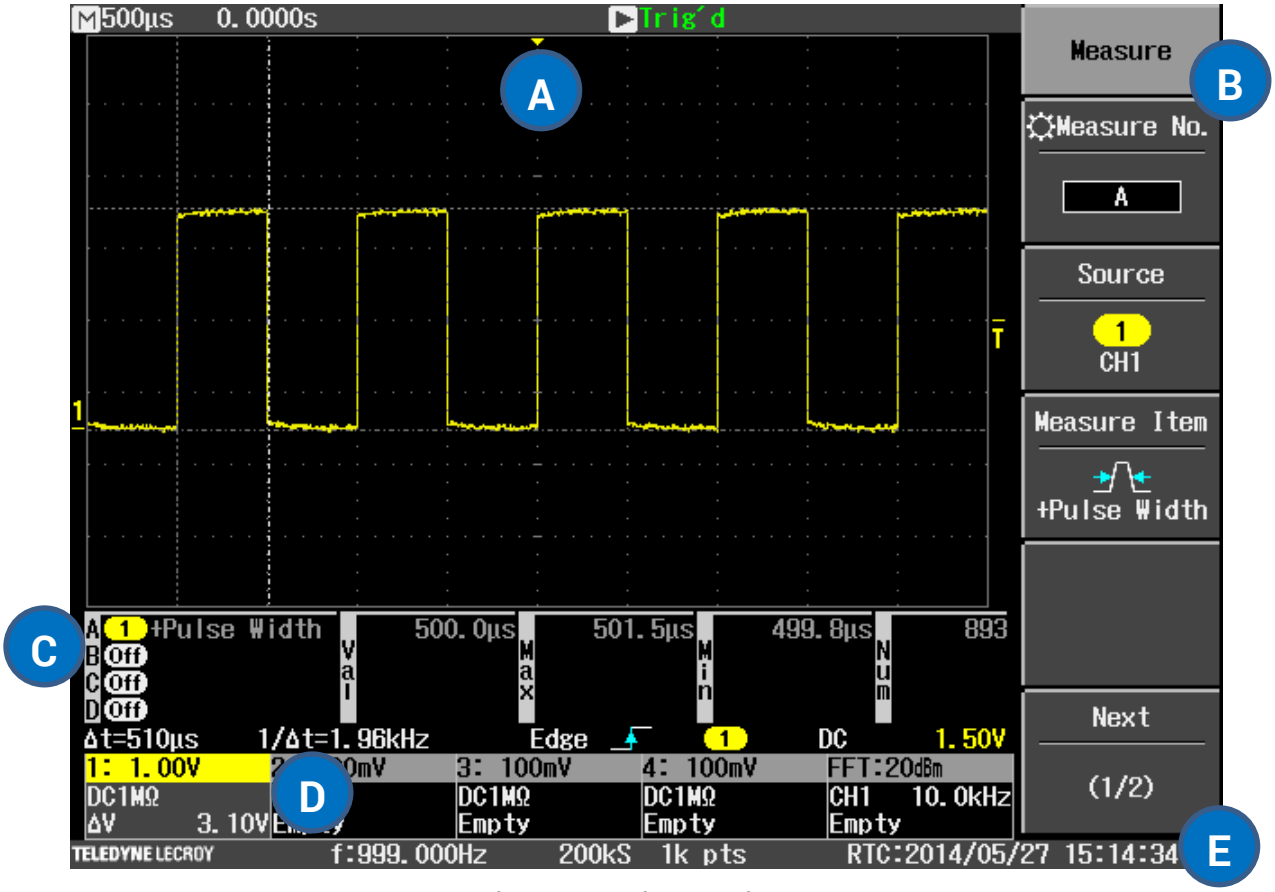


Figure 3: Touch screen layout

A. Grid

The grid displays the waveform traces. Grid indicators are described in the Channel, Timebase, and Trigger sections.

B. Touch Menus

Various setup menus appear to the right of the display when front panel buttons are pressed. These menus are touch activated.

The top cell is the menu name. On Channel and Math setup menus, it is color-coded to match the trace. Touch it to close the menu and return to the previous display.

The appearance of menu options indicates what type of entry is required:

- Icons or words set in a black box indicate a toggle. Continue touching the cell until the correct choice is selected. A white box surrounds the selected option.
- Labels indicate a sub-menu will open on top of the primary menu for further selection. Make a selection from the sub-menu to return to the previous menu in the stack. If the sub-menu remains open, touch the top cell or press the CLOSE MENU button on the front panel.

- The gear symbol indicates the ADJUST knob can be used to change the displayed value. The icon changes to when ADJUST is set to coarse (stepped increments).

Menus with Next (#/#) in the bottom cell have additional pages of options not yet displayed. Touch Next to see the next page of options. Continue touching Next to return to the top page.

C. Measurement Table

When the oscilloscope is in Measure mode (measurements turned on), a four-line measurement readout table is added to the display. This compresses the grid area. Turning off measurements restores the full trace display.

D. Descriptor Boxes

Channel and Math traces have a corresponding descriptor box below the grid, which shows the current settings on that trace. When the trace is off, the header is grey. When the trace is on, the header is color-coded to match the trace, and the descriptor box is touch enabled. Use it to open the CH or Math setup menu.

E. Status Bars

Acquisition Status Bar

The Acquisition Status bar appears at the top of the touch screen. It shows the current timebase, trigger delay, and the triggering status (Ready, Stop, etc.).

Figure 4: Acquisition status bar running

When acquisition is stopped, the Acquisition Status bar shows the Sweeps Count (#/#) at the far right, preceded by the Replay icon indicating the oscilloscope is now in Replay mode.

Figure 5: Acquisition status bar stopped

Trigger Status Bar

The Trigger Status bar appears below the grid. It summarizes the trigger settings and provides cursor measurement readout when cursors are enabled.

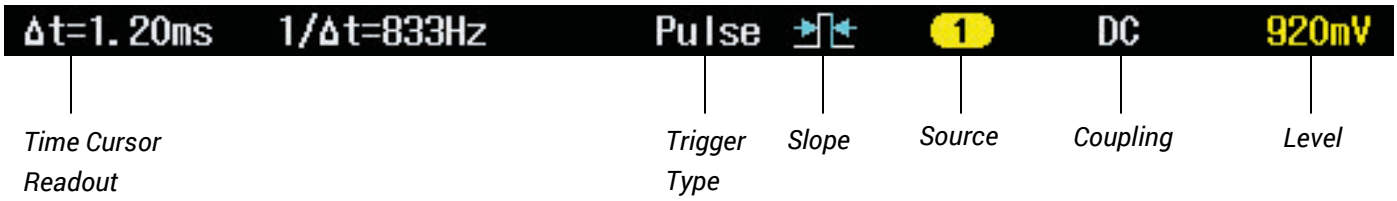


Figure 6: Trigger status bar

Message Bar

This Message bar appears at the bottom of the touch screen. It displays any processing messages (e.g., Pass/Fail results, errors) and shows the timestamp.

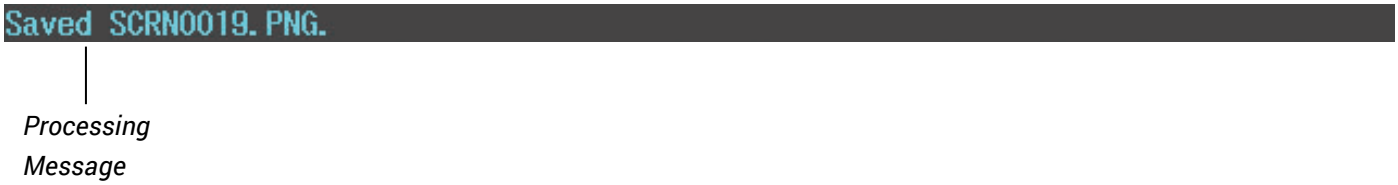


Figure 7: Message bar with processing message

NOTE: Status bar messages are generally low priority and disappear after a few seconds. High priority messages are displayed in pop-ups over the center of the display.

During normal operation when there is no message, it displays sampling information. The message bar can be configured to show the real-time clock or the acquisition timestamp.

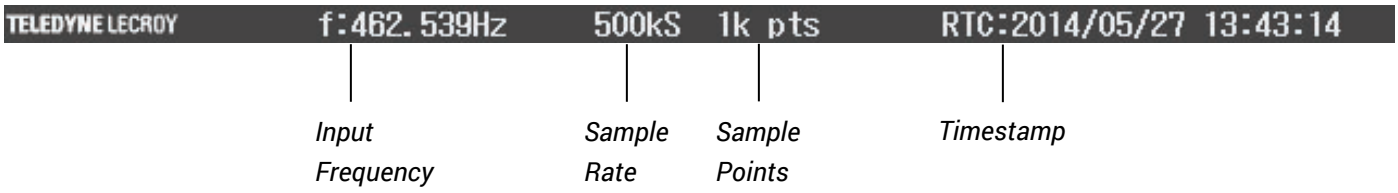


Figure 8: Message bar during normal operation

Setting Up the Oscilloscope

Carrying 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 disconnect all input devices and unplug the instrument from the power source before lifting and carrying it.

Placing the Oscilloscope

Secure a location for the instrument that is:

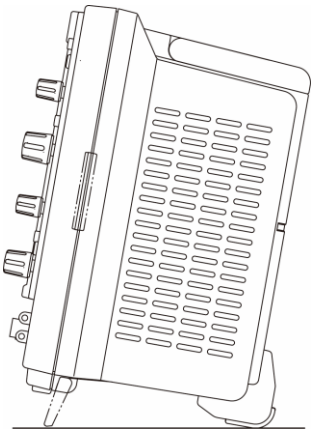
- **Stable**. Place the oscilloscope in a location that is flat and stable. If the installation site is not stable, the oscilloscope may fall over, possibly causing injury and/or damage to the instrument.
- **Well-ventilated**.
- **Maintained within the operating environment**. See Operating Environment.

CAUTION. Condensation may occur when moving this product from an environment with low temperature and humidity to one with high temperature and humidity, or when the equipment is otherwise exposed to rapid changes in temperature. In such cases, raise the ambient temperature gradually, and allow the equipment to adjust to the ambient temperature before using it.

The oscilloscope has ventilation holes and air exhaust outlets on both sides. To prevent the internal temperature from rising excessively, maintain a 15 cm (6-inch) distance from other objects so as not to inhibit the flow of air 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 two front legs can be pulled open to provide a tilt (about 10°) to enable easier viewing.

Powering On/Off

The power switch is located on the front of the oscilloscope in the lower left corner. Be sure the Power button is in "Standby" position (up) before plugging in the power cord. Press the button once to turn ON the oscilloscope. A few seconds after switching ON, a waveform trace will appear on the display.

NOTE: For best precision measurements, allow a warm-up period of at least 20 minutes after turning on the power.

Press the button again to turn OFF (Standby mode). The Power button does not completely disconnect the oscilloscope from the power supply. Some internal "housekeeping" circuitry continues to draw power. The only way to fully power down the instrument is to unplug the power cord from the outlet.

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

The most recent setups are retained in memory when the oscilloscope is powered down (Standby mode or unplugged). These setups are automatically loaded from memory after the next power-on.

NOTE: Auto Calibration settings are not retained. Auto Calibration is turned ON whenever the oscilloscope is powered on, even if you turned off this feature when last powering off.

Selecting Language

If desired, change the User Interface language after powering on. The selected language appears on all touch screen elements and in the online Help manual. See p.67.

Adjusting Trace and Grid Intensity

Set the intensity (brightness) of the waveform traces and the grid lines. See p.64. and p.69.

Setting Date and Time

Set the internal clock. See p.67.

Phase Correction of Probe

Phase correction adjusts the capacity of the probe's internal frequency to set constant gain. This adjustment must be performed correctly for accurate measurements. Be sure to perform phase correction when:

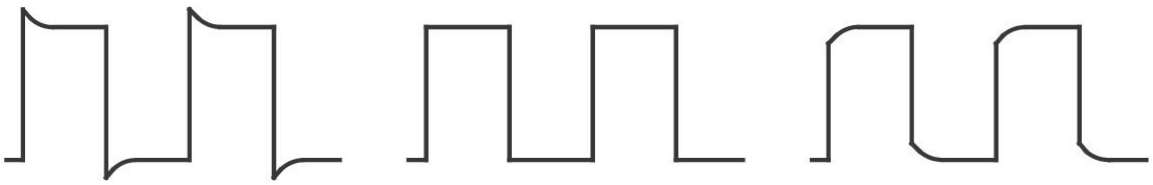
- Using any probe for the first time
- When connecting the probe to a different channel (as input capacitance varies slightly among channels even within the same oscilloscope)
- After reconfiguring or recalibrating the oscilloscope

The phase correction procedure utilizes the oscilloscope's CAL output signal, which is a 1 kHz, 3.0 VP-P square wave.

CAUTION: Do not apply a voltage from an external source to the CAL signal terminal. Doing so may damage the internal circuitry.

To perform phase correction:

1. **Power ON** the oscilloscope.
2. Connect the probe BNC connector to an input channel (**CH1 to CH4**).
3. Connect the probe tip to the **CAL terminal** and the ground lead to the **GND terminal**.
4. Press the **AUTO SETUP button**.
5. While viewing the square wave trace, insert the tip of the supplied screwdriver into the probe phase adjustment slot and turn it until the trace approximates an ideal square waveform: a flat top with as little overshoot or undershoot as possible.



Over compensated *Correct adjustment* *Under compensated*

Figure 9: Probe phase correction

Vertical (Channel)

Turn On Channel Trace

Connect your signal to the CH1-CH4 inputs, then press the corresponding **CH button** in the Vertical section of the front panel. The CH button lights, the **CH touch menu** opens, and the channel descriptor box changes from grey to the trace color to indicate the channel is now "on."

Vertical Readout

Channel Descriptor Box

Channel descriptor boxes below the grid show the current Vertical settings.

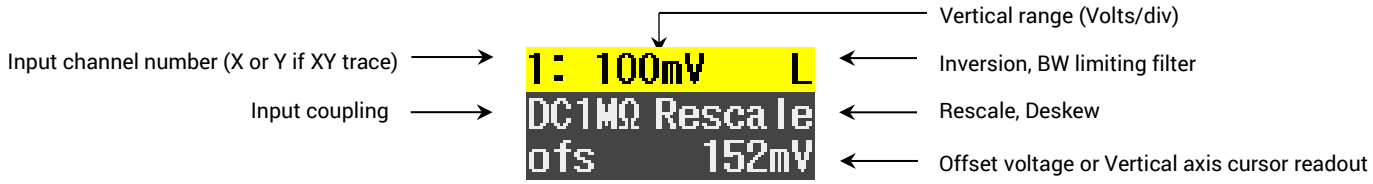


Figure 10: Channel descriptor box when VT trace is on

Table 1: Channel Descriptor Box Elements

Setting	Screen Display	Meaning
Range (V/div)	*nnn u*V | *nnn u*A *nnn u*W | *nnn u*C	Volts per Vertical division. If units changed, Amperes, Watts, or degrees Celsius per Vertical division.
Invert	[Blank]	Off (normal display)
	↓	Waveform inverted
Bandwidth	[Blank]	Full bandwidth
	BW	Bandwidth limiting on
	L	Low-pass filter on
	H	High-pass filter on
	S	Simple Moving Average on
Rescale	Rescale | Res	Rescaling on. Short form used when Deskew also on.
Deskew	Deskew | Des	Deskew on. Short form used when Rescale also on.
Offset	ofs *n.nn u*V	Offset voltage level
Coupling	AC1MΩ | GND DC1MΩ | DC50Ω	Coupling set to type shown

When Vertical cursors are applied to the channel, the Vertical Offset voltage is replaced by the cursor readout. See the Cursors section.

Whenever the channel is turned on (and the CH button lit), touching the descriptor box opens the CH menu. When the channel is off, the box is grey and inactive, although it will still display the channel settings.

Grid Indicators

The Vertical Offset voltage level is marked on the X axis by the channel number. It only appears when the channel is turned on. Indicators will overlap if multiple channels are set to Zero Offset.

Adjust Vertical Scale

Each channel has its own set of Vertical scale controls on the front panel.

Vertical Offset Knob

With the channel turned on, turn the Vertical **OFFSET knob** clockwise to move the trace in the positive direction (up) and counter-clockwise to move in the negative direction (down). Press OFFSET once to move the waveform display to OV in the target range. Press OFFSET again to restore the previous Offset setting.

Offset variation ranges are listed in Table 2. The offset variation range differs according to the Volts/div setting.

Table 2: Vertical Offset Range

Volts/div	Offset Range	
	Voltage Value	Div Value
10 V	–100 V to 100 V	–10 div to +10 div
5 V		–20 div to +20 div
2 V		–50 div to +50 div
1 V		–100 div to +100 div
500 mV	–10 V to +10 V	–20 div to +20 div
200 mV		–50 div to +50 div
100 mV		–100 div to +100 div
50 mV	–1 V to +1 V	–20 div to +20 div
20 mV		–50 div to +50 div
10 mV		–100 div to +100 div
5 mV		–200 div to +200 div
2 mV		–500 div to +500 div

Volts/div Knob

Turn the **Volts/div knob** clockwise to move toward a higher sensitivity range, or counter-clockwise to move toward a lower sensitivity range.

The Vertical axis range is from 2 mV/div to 10 V/div. If the probe's damping ratio is set to any value other than 1:1, the variation shown on the channel descriptor box shifts to 10x, 100x, 1000x, etc. (according to the probe's damping ratio).

Normally, this switch uses steps 1, 2, and 5, but the conversion setups are followed unless a conversion function has been set in the CH menu.

If the acquisition has been stopped and the Volt/div setting has been changed, pressing the **Volts/div knob** once will restore the previous setting.

Volts/div Increment

By default, Volts/div increments in coarse (1, 2, 5) steps. To make fine adjustments:

1. Touch the **channel descriptor box** to open the CH menu.
2. Touch **Next** to go to Page 2.
3. Touch **Volts/div** until Fine is selected.

Enlarge/Shrink Waveform

How a waveform appears to enlarge or shrink on the grid when Volts/div is adjusted depends on the Offset Setting constant selected on the Utilities > Config. menu:

- Division enlarges/shrinks the waveform relative to its own GND position, which does not change position.
- Volts enlarges/shrinks the waveform relative to center of screen (point 0 on the grid). Parts of the waveform that "disappear" off the grid when the Vertical range is increased can be viewed by adjusting the Vertical Offset.

Coupling

To change the input coupling:

1. Touch the **channel descriptor box** to open the CH menu.
2. Touch **Coupling** to open the Coupling sub-menu.
3. Choose from **AC1MΩ**, **GND**, **DC1MΩ**, or **DC50Ω**.

Bandwidth Limiting and Digital Filtering

A variety of filters may be applied to limit the input bandwidth.

1. Touch the **channel descriptor box** to open the Channel setup menu.
2. Touch **Bandwidth** to open the Bandwidth sub-menu.
3. Touch **BW Limit** and choose from **Full**, **100 MHz**, **20 MHz**, **2 MHz**, or **200 kHz**.
4. Touch **Filter** and choose from:
   - **OFF** (default) disables digital filtering.
   - **LPF** (Low-Pass Filter) attenuates and removes samples above the Cutoff frequency.
   - **HPF** (High-Pass Filter) attenuates and removes samples below the Cutoff frequency.
   - **SMA** takes a Simple Moving Average using the formula:
     Waveform data *n* = (waveform data (*n*-<*width*>) + waveform data *n* + waveform data (*n*+<*width*>)) / 3
     The **Width** setting determines the spacing between samples used to calculate the average. It can be set from ±1 to ±25.
5. If you chose LPF or HPF, touch the **Cutoff** cell that now appears on the Bandwidth menu, then turn the **ADJUST knob** until the desired cutoff frequency is displayed.

   If you chose SMA, touch the **Width** cell that now appears on the Bandwidth menu, then turn the **ADJUST knob** until the desired sampling width is displayed.

Cutoff Frequency Range

The available range and resolution of the Cutoff frequency depends on the sampling rate:

Lower limit of cutoff frequency ：sampling rate × 0.040［Hz］

Higher limit of cutoff frequency ：sampling rate × 0.460［Hz］

Resolution of cutoff frequency ：sampling rate × 0.001［Hz］

The setting range of the cutoff frequency depends on the sampling frequency. Therefore, the same time range is different on the setting range according to the memory length.

Actually, the cutoff frequency set by operating the ADJUST switch is preserved as an internal value. When the sampling rate is changed, it is likely to be limited to the lower limit value or the upper limit value within the range where the cutoff frequency can be set. However, the cutoff frequency returns to the preserved value when the sampling rate is changed again and the preserved value becomes within the range where the preserved cutoff frequency can be set. Moreover, when Recall Default Setup is executed by the SAVE/RECALL menu, the cutoff frequency is initialized by the lower limit value.

Exclusions

Bandwidth limiting/digital filtering is automatically turned off when any of the following occur:

- Peak Detect or Average acquisition modes are turned on
- Real memory length is less than 500 points
- Roll Mode is turned on
- Equivalent Sampling (Equ) is turned on

Probe

Set Attenuation

The Probe sub-menu enables you to configure the attached probe's Attenuation ratio.

1. Touch the **channel descriptor box** to open the Channel setup menu.
2. Touch **Next** to display page 2.
3. Touch **Probe**.

Save Setups

Also use the Probe sub-menu to save channel settings, which you can later recall to any other channel. This is a quick way to copy channel setups.

1. Touch the **descriptor box** of the channel whose setup you wish to save.
2. Touch **Next** to display Page 2.
3. Touch **Probe**, then touch **Save Probe Setup**.
4. Touch the location in which to save the setup, **Probe Setup1** through **Probe Setup4**. The current configuration will overwrite what is now saved in that location.

Recall Setups

To recall saved setups:

1. Touch the **descriptor box** of the channel to which you want to copy a saved setup.
2. Touch **Next** to display Page 2.
3. Touch **Probe**, then touch **Recall Probe Setup**.
4. Choose the setup to recall, **Probe Setup1** through **Probe Setup4**. The saved configuration is copied to the channel and appears on the channel descriptor box.

When a Probe Setup is recalled the offset setting is persevered.

NOTE: After a firmware upgrade, Probe Setup1 through Probe Setup4 revert to the default channel settings (V Unit, Auto Attenuation, Full Bandwidth, DC1MΩ Coupling).

Invert Waveform

Invert displays the waveform inverted about the center of the screen, regardless of the Vertical axis range setting.

1. Touch the **channel descriptor box** to open the CH menu.
2. Touch **Invert** until **On** is selected.

NOTE: Invert is set independently for each channel. When Invert is on, all Math, Auto Measure and Cursor measurements are calculated in relation to the inverted waveform. Waveforms that are inverted when saved using the SAVE/RECALL feature are inverted when recalled to the display.

Units

To change the Vertical axis units:

1. Touch the **channel descriptor box** to open the CH menu.
2. Touch **Next** to display Page 2.
3. Touch **Unit**.
4. Choose from **Volt** (default), **Ampere**, **Watt**, **Degrees C**, or **No Unit**.

Rescale

This function allows you to change the Vertical axis scale by applying the formula ax+b, or: <vertical range> × <input voltage> + <vertical offset>. It can be especially useful when measuring current.
To rescale a trace:

1. Touch the **channel descriptor box** to open the CH menu.
2. Touch **Next** to open Page 2.
3. Touch **Rescale** to open the Rescale sub-menu.
4. Touch **Rescale (ax+b)** to select **On**.
5. Touch **a** to select the first field, then turn the **ADJUST knob** until the desired value appears.
6. Touch **a** again to select the second field, then turn the **ADJUST knob** until the desired value appears
7. Touch **b** to select the field, then turn the **ADJUST knob** until the desired value appears.

NOTE: When rescaling is turned on, “Rescale” or “Res” appears next to the coupling on the channel descriptor box. Rescaling a trace may modify the readout of Math operations applied to it. See Math Readout.

Deskew

Deskew sets a time differential to correct for the inter-channel phase differential. For example, it can be used to adjust for propagation delays due to differences among probes/cables.

NOTE: The available range of the Deskew function depends on the Horizontal Preference setting. For maximum range (0-500 ns) at all timebases, make Deskew the preference.

1. Touch the **channel descriptor box** to open the CH menu.
2. Touch **Next** to open Page 2.
3. Touch **Deskew** to select the field, then turn the **ADJUST knob** until the desired value appears.

NOTE: When Deskew is applied, “Deskew” or “Des” appears next to the coupling in the channel descriptor box.

Auto Setup

The Auto Setup feature uses the characteristics of the input signal to automatically set Vertical range (V/div), timebase (Time/div), and trigger level. It is useful for displaying waveforms when the setup requirements are not known because it is unclear what kind of signal it is.

Press the front panel **AUTO SETUP button** after turning on channels using the front panel CH buttons. Auto Setup will be performed on all active channels.

NOTE: If a channel is not itself currently turned on, but is referenced by another active trace—such as the source channels of a Math trace—the Auto Setup function will turn on and configure those channels.

To restore to the prior channel setups, touch **Undo** on the Auto Setup menu.

CAUTION. Operating any controls after entering Auto Setup will exit Auto Setup and void the ability to return to the previous setups using the Auto Setup menu Undo option.

Restore Default Setup

Use the Save/Recall function to restore the factory default channel setups. See p.72.

XY Traces

Channel traces may be displayed in VT or XY format. See Display.

Trigger

The trigger type determines the event that will be used to start the acquisition. When this event occurs at the specified trigger level, the acquisition begins.

Trigger Readout

Acquisition Status Bar

Table 3: Acquisition Status Bar Trigger Elements

Setting	Screen Display	Meaning
Trigger Mode	Auto	AUTO trigger enabled
	Ready	(NORMAL or SINGLE) trigger ready
	Trig'd	(NORMAL or SINGLE) trigger fired
	Inhibit	Triggering inhibited
	Stop	Triggering stopped

Trigger Status Bar

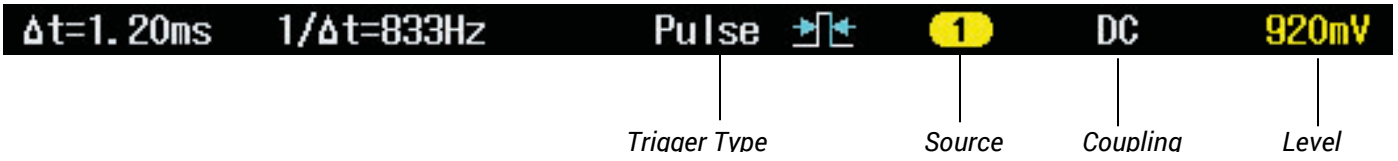


Table 4: Trigger Status Bar Trigger Elements

Setting	Screen Display	Meaning
Trigger Type	Edge	Edge trigger set
	Edge ALT	Edge alternate trigger set
	Edge OR	Edge OR trigger set
	Pulse Count	Pulse Count trigger set
	Pulse Width	Pulse Width trigger set
	Period	Period trigger set
	Dropout	Dropout trigger set
	TV	TV trigger set
	OR	OR (Logical Sum) trigger set
	NOR	Negative OR trigger set
	AND	AND trigger set

Setting	Screen Display	Meaning
	NAND	Negative AND trigger set
	SPI UART I2C	Serial trigger set
Slope (Edge triggers)		Positive slope
		Negative slope
		Alternate slopes (Edge ALT trigger only)
		Don't Care (Edge OR trigger only). Indicates the channel is in effect excluded from the pattern.
Source	1, 2, 3, 4	Input channel number. For Serial triggers, the number is followed by DAT, CLK or CS indicating it is the Data, Clock, or Chip Select signal source.
	Ext, Ext/10	External trigger source used. For Serial triggers, letter E is followed by DAT, CLK or CS indicating it is the Data, Clock, or Chip Select signal source.
	Line	Available only for Edge trigger. It generates a trigger according to the period of the AC power supply in use.
TV Mode	NTSC | PAL | Custom	TV trigger source signal format
Level	± *nnn u*V	Triggering voltage. This number is positive when the Trigger Level is higher than the Vertical Offset, and negative when it is lower.
Field	Any Field, 1, 2, 3, 4, 5, 6, 7, 8	Field in frame/message/packet to watch for trigger event (Serial trigger only)

Grid Indicators

Trigger Level is marked by a T on the Vertical axis. The T is always the same color as the trigger source channel. If the level is above (positive to) the Offset Level, a line appears above the T.

If the level is below (negative to) the Offset Level, a line appears below the T.

Adjust Trigger Level

The **LEVEL knob** sets the trigger level for all channels. Turn the knob clockwise to trigger at a higher level, and turn it counter-clockwise to trigger at a lower level. The Trigger Level readout on the Trigger Status bar will be positive or negative depending on whether this value is higher or lower than the Vertical Offset of the trigger source trace.

Press the LEVEL knob once to **FIND LEVEL**, which sets the trigger level as the center of the amplitude range of the trigger source channel waveform.

Set Up Trigger

The setup required for each different type of trigger is described below.

Edge Trigger

Edge is the default trigger type. It triggers when the trigger source waveform edge reaches the trigger level.

1. Press the **Trigger Setup button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then choose **Edge**.
3. Touch **Source** and choose the trigger source signal.
4. Turn the Trigger **LEVEL knob** to set the triggering level for the channel.
5. Touch **Slope** until the desired edge icon (positive or negative) is selected.
6. Touch **Coupling** and choose from AC, DC, HF Reject, LF Reject, and DC Noise Reject.
7. To apply holdoff time to the trigger, touch **Holdoff** and turn the **ADJUST knob** until the desired time appears.

Edge Alternate Trigger

Edge Alternate triggers alternately at the rising and falling edges of the trigger source waveform.

1. Press the **Trigger Setup button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then choose **Edge ALT**.
3. Touch **Source** and choose the trigger source signal.
4. Turn the Trigger **LEVEL knob** to set the triggering level for the channel.
5. Touch **Coupling** and choose from AC, DC, HF Reject, LF Reject, and DC Noise Reject.
6. To apply holdoff time to the trigger, touch **Holdoff** and turn the **ADJUST knob** until the desired time appears.

Edge OR Trigger

Edge OR triggers whenever any of the included waveforms edges meet the triggering conditions set. Multiple channels may be included in the trigger, or excluded by setting the Slope to “Don’t Care.” External and AC Line signals cannot be used with this trigger.

1. Press the **Trigger Setup button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then choose **Edge OR**.
3. Touch **Source**, then select **CH1**. Touch **Slope**, then choose to use the Positive or Negative edge of CH1, or exclude it from the trigger altogether by selecting “Don’t Care.”
4. Turn the Trigger **LEVEL knob** to set the triggering level for this channel.
5. Touch **Coupling** and select the coupling for this channel from AC, DC, HF Reject, LF Reject, and DC Noise Reject.
6. Repeat Steps 3-5 for CH2, CH3, and CH4.

NOTE: It is only necessary to set Coupling and Trigger Level for channels that are included in the trigger.

Pulse Count (Event) Trigger

Pulse Count triggers when the specified number of trigger signal pulses is reached. Pulse Count triggers are well suited for checking the operation of a counter circuit or encoder.

1. Press the **Trigger Setup button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then choose **Pulse Count**.
3. Touch **Source** and choose the trigger source signal.
4. Turn the Trigger **LEVEL knob** to set the triggering level for this channel.
5. Touch **Coupling** and select from AC, DC, HF Reject, LF Reject, and DC Noise Reject.
6. Touch **No. of Pulse** then turn the **ADJUST knob** until the desired pulse count is shown. The valid range is 1 to 9999.

Pulse Width Trigger

Pulse Width triggers when the trigger signal matches the specified polarity and pulse width conditions.

1. Press the **Trigger Setup button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then touch **Next** and choose **Pulse Width**.
3. Touch **Source** and choose the trigger source signal.
4. Turn the Trigger **LEVEL knob** to set the triggering level for this channel.
5. Touch **Polarity** until the desired polarity icon (Positive or Negative) is selected.
6. Touch **Coupling** and select from AC, DC, HF Reject, LF Reject, and DC Noise Reject.
7. Touch **Pulse Width**, then touch **When** and choose the formula that describes the triggering condition.
8. For each **variable** that now appears on the Pulse Width sub-menu (m and/or n), touch the field and turn the **ADJUST knob** until the desired value is shown in the field. This sets the width condition that must be met to fire the trigger.

Period Trigger

Period triggers when conditions are met within a certain signal period. It can be used for pulse measurements, such as in motors.

1. Press the **Trigger Setup button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then touch **Next** and choose **Period**.
3. Touch **Source** and choose the trigger source signal.
4. Turn the Trigger **LEVEL knob** to set the triggering level for this channel.
5. Touch **Slope** until the desired edge icon (Positive or Negative) is selected.
6. Touch **Coupling** and select from AC, DC, HF Reject, LF Reject, and DC Noise Reject.
7. Touch **Interval Time**, then touch **When** and choose the formula that describes the triggering condition.
8. On the Pulse Width sub-menu, touch **m** and turn the **ADJUST knob** until the desired value is shown in the field. This sets the period condition that must be met to fire the trigger.

NOTE: The Period trigger only operates within certain frequencies. The upper limits per period range are shown in Table 5 below. Operation is not guaranteed if the frequency is over this limit.

Table 5: Operational Frequencies for Period Trigger

Period	Max. Operating Frequency
to 167.6 msec	40 MHz
to 167.6 msec	40 MHz
to 335.2 msec	20 MHz
to 670.4 msec	10 MHz
to 1.3 sec	5 MHz
to 1.6 sec	4 MHz
to 3.3 sec	2 MHz
to 6.7 sec	1 MHz
to13.4 sec	500 kHz
to16.7 sec	400 kHz
to 33.5 sec	200 kHz
to 50.0 sec	100 kHz

Dropout Trigger

Dropout triggers if the specified edge was not detected during the dropout time. Dropout can be used to check whether pulses are being generated as per specifications.

1. Press the **Trigger Setup button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then touch **Next** and choose **Dropout**.
3. Touch **Source** and choose the trigger source signal.
4. Turn the Trigger **LEVEL knob** to set the triggering level for this channel.
5. Touch **Slope** until the desired edge icon (Positive or Negative) is selected.
6. Touch **Coupling** and select from AC, DC, HF Reject, LF Reject, and DC Noise Reject.
7. Touch **Dropout Time**, then turn the **ADJUST knob** until the desired value is shown in the field. This sets the dropout condition that must be met to fire the trigger.

TV Trigger

TV triggers when the specified TV signal field or line meets the edge and frequency conditions.

1. Press the **Trigger Setup button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then touch **Next** and choose **TV**.
3. Touch **Source** and choose the trigger source signal.
4. Touch **Slope** until the desired edge icon (Positive or Negative) is selected.
5. Touch **TV Setting**, then touch **TV Standard** and select the signal type from **NTSC**, **PAL**, or **Custom**.

NTSC and PAL Signals

6. Touch **Field Sequence** and turn the **ADJUST knob** until the number of fields in the signal appears.
7. Touch **Field No.** and turn the **ADJUST knob** until the field on which to trigger appears
8. To use any line in the field, touch the bottom cell until **Any Lines** in selected.

   To use a specific line, touch the bottom cell until the **number** is selected, then turn the **ADJUST knob** until the desired line number appears.

Custom Signals

6. Touch **Custom Setting**, then touch **Field Frequency** and turn the **ADJUST knob** until the correct number of Hz is shown.
7. Touch **No. of Scan Line** and turn the **ADJUST knob** until the correct number of lines is shown.
8. Press **Close Menu** to return to the TV Setting menu.
9. Touch **Field Sequence** and turn the **ADJUST knob** until the number of fields in the signal appears.
10. Touch **Field No.** and turn the **ADJUST knob** until the field on which to trigger appears
11. To use any line in the field, touch the bottom cell until **Any Lines** in selected.

    To use a specific line, touch the bottom cell until the **number** is selected, then turn the **ADJUST knob** until the desired line number appears.

Pattern Triggers (OR, NOR, AND and NAND)

These triggers allow you to define a High or Low voltage threshold level on one or more channels and fires when the source signals are in the High or Low states that match the trigger pattern.

1. Press the **Trigger Setup button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then touch **Next** until you see page 3/4.
3. Choose the Type of:
   - OR —fires when any one of the threshold conditions are met.
   - NOR —fires when none of the threshold conditions are met.
   - AND —fires when all the threshold conditions are met.
   - NAND —fires when at least one of the threshold conditions isn't met.
4. Touch **Source**, then select **CH1**. Turn the Trigger **LEVEL knob** to set the trigger level for this channel.
5. Touch **State**, then choose to trigger when CH1 is above the level (High) or below it (Low). To exclude it from the trigger conditions altogether, select "Don't Care."
6. Touch **Coupling** and select the coupling for this channel from AC, DC, HF Reject, LF Reject, and DC Noise Reject.
7. Repeat Steps 4-6 for CH2, CH3, and CH4.

UART Serial Trigger

UART triggers are used to monitor waveforms from start/stop synchronization type serial communications, such as RS-232C communications or the UART (Universal Asynchronous Receiver Transmitter) serial communications standard.

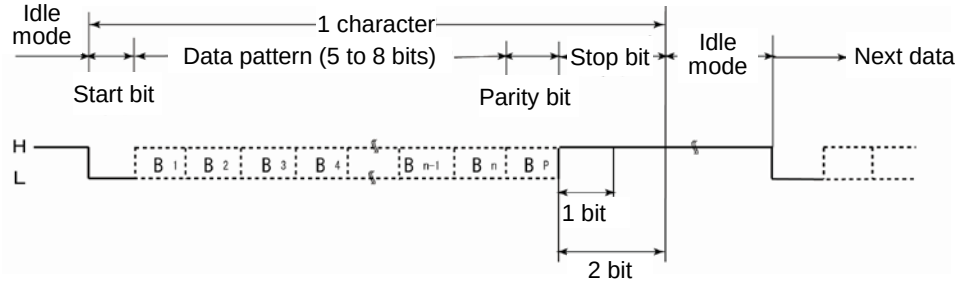


Figure 11: UART protocol

The UART trigger may be set on the:

- **Start bit** or **Stop bit**
- A specific **Data pattern**. Bits in the received data are compared to the data pattern bit string sequentially from the left, and a trigger occurs when all the bits match.
- A **Parity** mismatch (in signals with a Parity bit only)

To set up a UART trigger:

1. Press the **Trigger SETUP button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then touch **Next** until you see Page 4/4.
3. Touch **UART**, then touch **Trigger** and choose to trigger on the Start bit, Stop bit, Data pattern, or Parity Error.
4. Touch **UART Setting** and enter the signal parameters:
   - Choose the input signal **Source** channel Turn the Trigger **LEVEL knob** to set the signal threshold level. It will appear under Source on the UART Setting menu.
   - Touch **Coupling** and choose the signal coupling type. Normally, DC should be selected.
   - Touch **Bit Rate**, then turn the **ADJUST knob** to enter the signal bit rate, from 1000 to 1.000 Mbps (in 100 bps units).
   - Touch **Num Bits**, then turn **ADJUST** to enter the length of the Data field, from 5 to 8 bits. Touch **Next**.
   - Touch **Stop Bit**, then turn **ADJUST** to enter the number of Stop bits, 1 or 2.
   - Touch **Parity** and choose the signal parity: None, Even, or Odd.

     NOTE: In order to set the triger type to Parity error, first set Parity to even or odd.
   - Touch **Idle Pol** and choose the idle mode polarity: Low level or High level.
   - Touch **Bit Order** and choose LSB (least significant bit) or MSB (most significant bit).

TIP: The bottom cell on the UART menu is an option to Show Chart. Touch this cell at any time to see the current state of the trigger pattern you have set up on all signals.

Data Pattern Triggers

5. Press **CLOSE MENU** to return to the Trigger menu.
6. Touch either the **Data** cell to enter the bit string, or the **Data (HEX)** cell to enter the byte unit.

   NOTE: The fields are linked. As you adjust one, you will see the other value entered in the respective format.
7. **Turn ADJUST** to enter the first digit/character, then **push ADJUST** to move to the next place.
8. Repeat until all values are entered.

SPI Serial Trigger

The SPI trigger is used to monitor signals from a SPI (Serial Peripheral Interface). Either a 3-channel (with chip select) or a 2-channel signal maybe input. A timeout is used when the chip select signal is absent. Once the clock has been in idle mode for the time specified by timeout, the first clock to be detected becomes the start of the frame. A trigger occurs when the pattern specified for bits 4 to 64 is received. In the examples below, the trigger pattern entered is found in byte N. Either the pattern was not found in earlier bytes, or they have been excluded from the trigger.

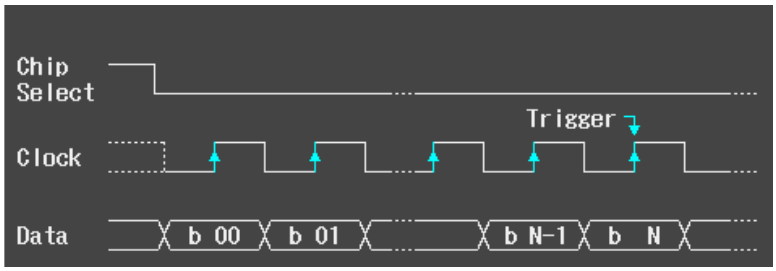


Figure 12: Input of 3-channel SPI signal for chip select, clock, and data.

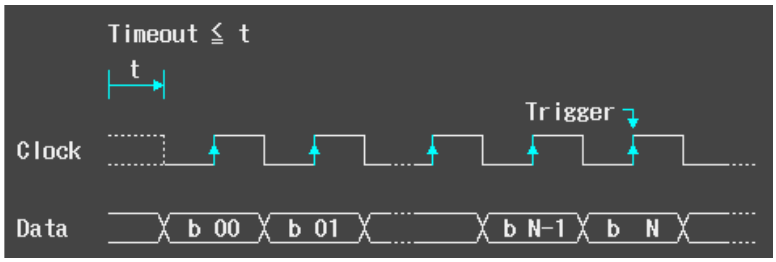


Figure 13: Input of 2-channel SPI signal for clock and data.

SET UP SIGNALS

1. Connect the Clock signal to CH1 (only).
2. Connect the Data signal to any remaining channel or to the EXT input.
3. If using a Chip Select line, connect the signal to any remaining channel or to the EXT input.

Enter Trigger Pattern

1. Press the **Trigger SETUP button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then touch **Next** until you see Page 4/4.
3. Choose **SPI**.
4. Touch **Num Bits**, then turn **ADJUST** to enter the total number of bits in the Data signal, from 4 to 64.
5. Touch the **Data** cell and begin to enter the pattern for the first byte (b00 – b07):
   - With the first bit position selected, **turn ADJUST** until the correct digit appears. To exclude this bit from the trigger pattern, enter an "X" ("Don't Care").
   - **Push ADJUST** to move to the next bit, then **turn ADJUST** to set the correct digit. Continue until all bits are adjusted.

     TIP: To quickly fill the entire pattern with the selected digit, touch Set All Bits. This saves you the work of having to manually adjust every bit in bytes that are to be completely excluded from the trigger pattern (all "X"), or in long patterns that are predominantly a 1 or 0.
   - Whenever the Num Bits setting is over 8, the byte number field inside the Data cell is also active, and pushing ADJUST will eventually highlight this field. Turn ADJUST to go to the next byte that requires adjusting. Push ADJUST again to go back to editing the bits.

Enter Signal Parameters

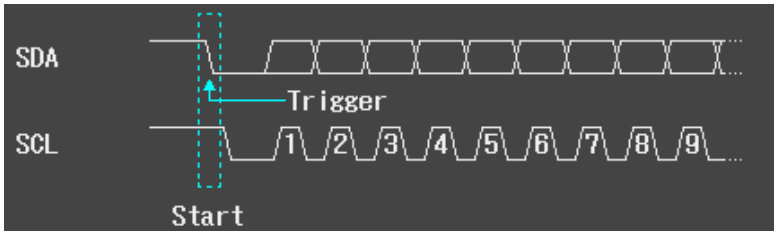
1. Touch **SPI Setting** to open the SPI Setting menu.
2. Touch **Signal** and choose **Clock**. Enter the Clock signal parameters:
   - Source will default to CH1. Turn the **Trigger LEVEL knob** to set the clock signal threshold. The value will appear on the Source cell of the SPI Setting menu.
   - Touch **Coupling** and choose the type. Normally DC should be selected.
   - Touch **Edge** and choose the clock signal detection edge.
3. Touch **Signal** and choose **Data**.
   - Touch **Source** and choose the data input. Turn the **Trigger LEVEL knob** to set the signal threshold.
   - Touch **Coupling** and choose the type.
4. Touch **Signal** and choose **Chip Select**. Enter the signal parameters:
   - Touch **Type**.
     - If using a 3-line signal, choose the polarity of the Chip Select line, Low level or High level.
     - If using a 2-line signal, choose Timeout.
   - If using a Timeout, touch **Timeout**, then turn the **ADJUST knob** to set the timeout value.
   - If using a Chip Select line:
     - Touch **Source** and choose the CS input. Turn the **Trigger LEVEL knob** to set the signal threshold.
     - Touch **Coupling** and choose the type.

TIP: The bottom cell on the SPI menu is an option to Show Chart. Touch this cell at any time to see the current state of the trigger pattern you have set up on all signals.

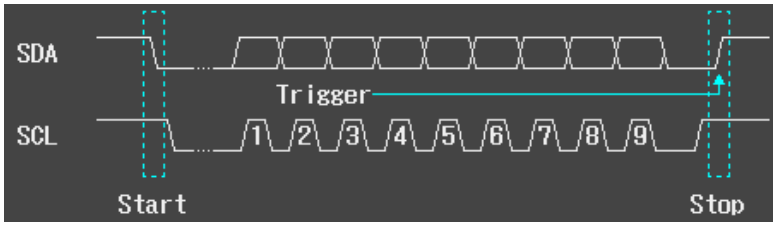
I²C Serial Trigger

I²C triggers are used to monitor I²C (Inter Integrated Circuit) signal waveforms. Two channels are input: SDA (data) and SCL clock. A number of different triggering conditions may be set:

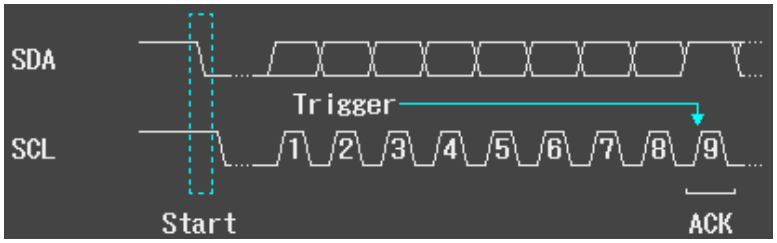
- **Start Bit** identified



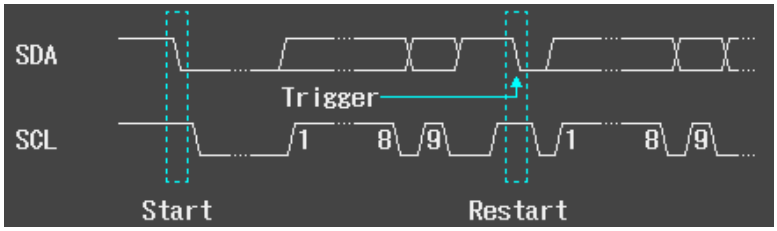
- **Stop Bit** identified



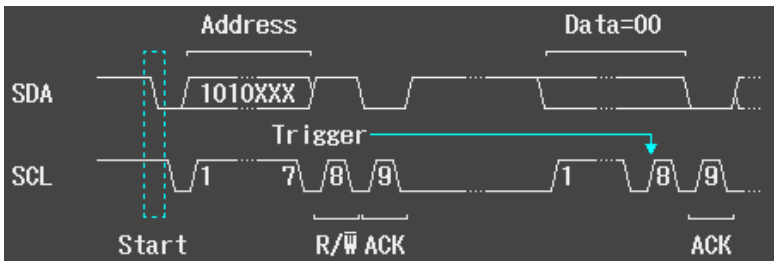
- **No ACK** (ACK bit goes to the high level)



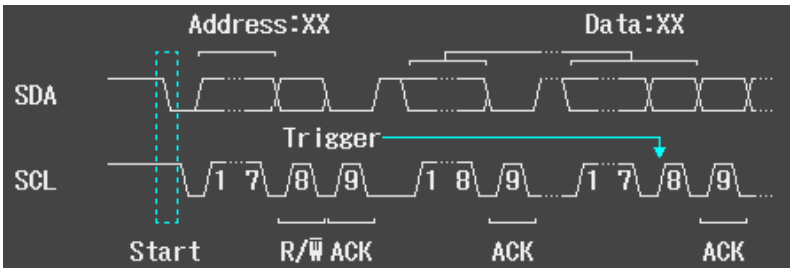
- **Restart** detected



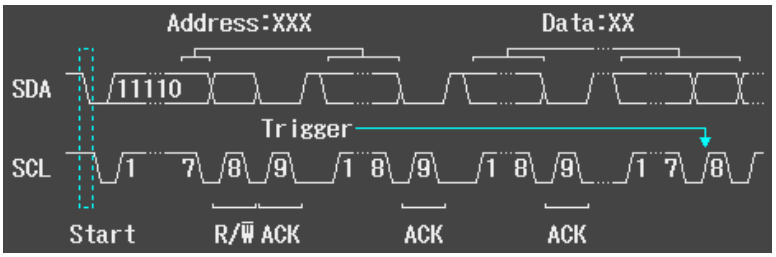
- **EEPROM Data Read** (data read from EEPROM meets the conditions when compared with 1 byte of data)



- **7bit Address & Data** (7-bit address, transfer direction, and data pattern in 1 to 5 bytes match conditions)



- **10bit Address & Data** (10-bit address, transfer direction, and data pattern 1 to 5 bytes match conditions)



To set up an I2C trigger:

1. Press the **Trigger SETUP button** to display the Trigger menu.
2. Touch **Type** to display the Type sub-menu, then touch **Next** until you see Page 4/4.
3. Touch **I2C**, then touch **Trigger** and choose one of the types described above.
4. Touch **I2C Setting** and enter the signal parameters:
   - Touch **Signal** and choose **SCL**.
   - Choose the clock signal **Source** channel. Turn the Trigger **LEVEL knob** to set the signal threshold level. It will appear under Source.
   - Touch **Coupling** and choose the signal coupling type. Normally, DC should be selected.
   - Touch **Signal** and choose **SDA**
   - Repeat setting the **Source** and **Coupling** for the data signal.
   - Address & Data triggers only: Touch **Data Length** and turn **ADJUST** until the number of bytes is shown.

TIP: The bottom cell on the I2C menu is an option to Show Chart. Touch this cell at any time to see the current state of the trigger pattern you have set up on all signals.

EPROM Data Read

If triggering on an EPROM data condition:

5. Press **Close Menu** to return to the Trigger menu.
6. Touch **Data**, then **turn ADJUST** to select an operator describing the trigger data condition.
7. **Press ADJUST** to go to the byte field, then enter the byte in hexadecimal, turning the ADJUST knob to select the first digit/character. Repeat for the second digit/character.

Address & Data

If triggering on Address & Data, enter the address, transfer direction and a data pattern of 1 to 5 bytes.

5. Press **Close Menu** to return to the Trigger menu.
6. Touch **Address** and turn **ADJUST** to select the transfer direction.
7. **Press ADJUST** and enter the address in hexadecimal. Turn ADJUST to select each digit/character; press ADJUST to move to the next position.
8. Touch **Data**, then turn **ADJUST** to choose the first byte to search for the trigger pattern.
9. **Press ADJUST** and enter the trigger pattern in hexadecimal, same as you did for the address.
10. Repeat Steps 8 and 9 to select more bytes to include in the trigger pattern. Continue pressing ADJUST to move to the next field, and turning ADJUST to change the value.

Trigger Mode

The Trigger Mode buttons on the front panel are used to control the acquisition.

AUTO button starts acquisition in Auto mode. The oscilloscope will trigger repeatedly when there is a valid trigger condition or after a preset period when there is no valid trigger condition.

NORMAL button starts acquisition in Normal mode. Waveform data is acquired each time a trigger condition is met.

SINGLE button starts a single-shot acquisition.

When the acquisition is paused in Auto or Normal mode, press the **RUN/STOP button** to restart acquisition.

Stop/Restart Acquisition

To stop the acquisition in progress, press the **RUN/STOP button**.

NOTE: A single-shot acquisition will automatically switch to Stop mode following the trigger firing (Trig'd). Press SINGLE and RUN again to repeat the acquisition.

To resume acquisition, press RUN/STOP again.

Horizontal (Timebase)

Horizontal settings control the timebase for all channels (Time/div), plus acquisition mode and memory length.

Horizontal Readout

Acquisition Status Bar

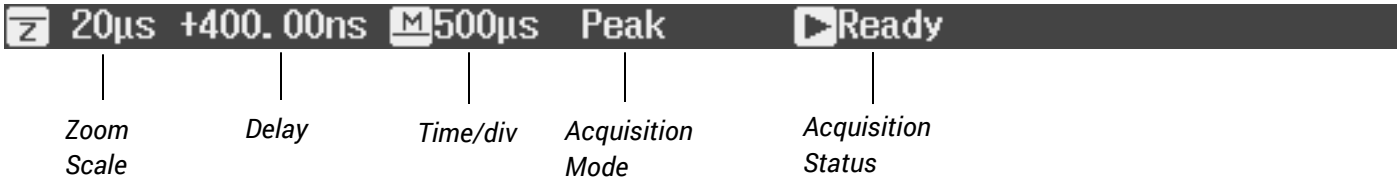


Table 6: Acquisition Status Bar Horizontal Elements

Setting	Screen Display	Meaning
Zoom Scale	<icon> *nnn uuu*	Zoom trace Horizontal scale (Time/div)
Delay	*nnn uuu*	Time (positive or negative) trigger moved from the 0 point of Y axis
Time/div	<icon> *nnn uuu*	Time represented by each division of the grid
Acquisition Mode	[Blank]	Normal Mode enabled
	Peak	Peak Detect Mode enabled
	Avg	Average Mode enabled
	HiRes	High Resolution enabled
Replay Mode		Shown only when acquisition is stopped. Indicates sweeps are available for Replay.
Sweeps Count	*nnnn/nnnn*	Shown only when acquisition is stopped. Current sweep number out of the total sweep number from the last acquisition.

Message Bar

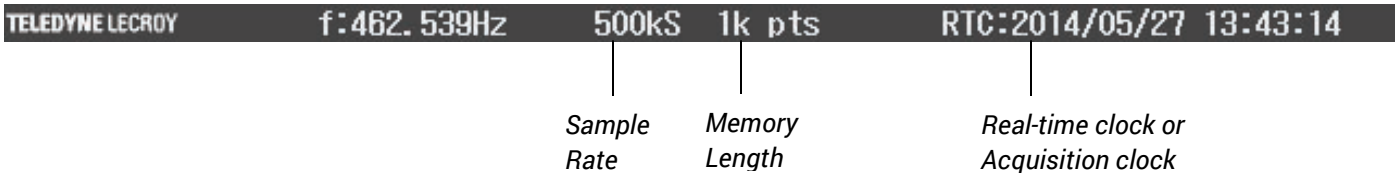


Table 7: Message Bar Horizontal Elements

Setting	Screen Display	Meaning
Sample Rate	*nnn uuu*	Samples per unit of time
Memory Length	*nnn u* pts	Total effective sample points
Clock	RTC *yyyy/mm/dd hh:mm:ss*	Real-time clock date and time
	ACQ *hh:mm:ss.n*	Acquisition time clock showing acquisition end time. Shown only when triggers are stopped. Real-time clock shown during the acquisition.

Grid Indicators


Horizontal Delay time is marked on the Y axis by as small triangle at the top of the grid. It is always displayed in the color of the trigger source channel.

Adjust Horizontal Scale

Horizontal **DELAY knob** adjusts the trigger delay for all channels. Turn the knob clockwise to move the waveform in the positive direction (right), and counter-clockwise to move it in the negative direction (left). When this knob is pressed once, the waveform is centered on 0 seconds. When the knob is pressed again, the previous Delay setting is restored. Delay times are shown in the Acquisition Status bar in the upper left part of the screen.

Time/div knob (Horizontal range) sets the timebase for all channels. Turn the knob clockwise to set a shorter timebase range, or counter-clockwise to set a longer timebase range. If the acquisition is stopped pressing the Time/div knob will restore the previous Time/div setting if the timebase has been altered since the acquisition has been stopped. The available range varies according to the model bandwidth and Max. Memory Length settings.

Table 8: Horizontal Range

	Wave Jet 354T	Wave Jet 334T
Frequency bandwidth	500 MHz	350 MHz
Time/div range[1]	500 ps/div to 50 s/div	1 ns/div to 50 s/div

1 Roll Mode operation begins at 50 ms/div. The maximum Horizontal range depends on the sampling rate. See Table 9: Sampling Rate and Vertical Axis Resolution.

Acquisition Mode

The Acquisition Mode determines how the oscilloscope samples waveform data. To select a mode:

1. Press the **Horizontal SETUP** button to open the Horizontal menu.
2. Touch **Mode** to display the Mode sub-menu.
3. Choose one of the following modes.

Normal

By default, the oscilloscope samples in Normal mode. Normal 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.

Peak Detect

Peak Detect mode detects the maximum value and minimum values that are issued at intervals equal to twice the sampling period used when Peak Detect is not set, and alternately records the results to memory. When using Peak Detect the sampling period is 1 ns. It is during this sampling period that the maximum and minimum values are detected.

NOTE: Being that the Peak Detect sampling period is 1 ns, Peak Detect does not operate if the sampling rate is 1 GS/s or above. If the sampling rate is too high, use the Time/div control knob to adjust it, or change the Max Memory Length in the Horizontal menu.

When using Normal acquisition, under some specific acquisition settings (sample rate and memory depth) it is not possible to detect waveform data point A in the figure below unless Peak Detect has been set. As is shown in the lower part, when Peak Detect mode is enabled, acquisition occurs in 1-ns cycle regardless of the sampling period, so any events that occur during the sampling period can be acquired.

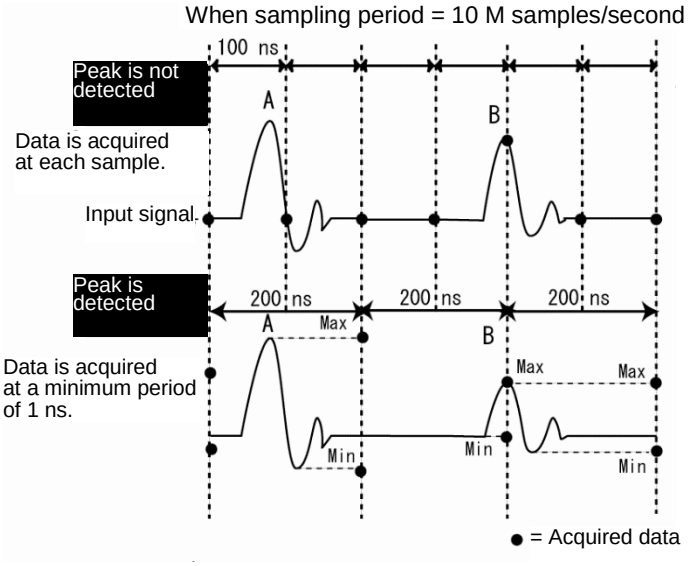


Figure 14: Peak Detect sampling at 10 M samples/second

Average

In Average mode, averaging is performed for data on the same time axis each time an input signal is acquired, and a waveform is displayed. The higher the averaged count value is, the more noise can be reduced in the observed signal, although processing time also increases.

When the acquisition is running, pressing the CLEAR button re-initializes averaging.

The calculation of the weighted average is as follows.

An = An−1+(Dn−An−1)/n (n ≤ N for all acquisition modes)

An = An−1+(Dn−An−1)/N (n > N in Normal and Auto modes)

Where:

n : Current averaging frequency (This is the value processed by the equipment. It cannot be displayed on screen or set by users)

N : No. of Times (This value is set by users)

An : Mean value of times n

Dn : Measurements of times n

Turning on Averaging does not limit the maximum memory length.

NOTE: Average acquisition mode is not compatible with Roll Mode display. If Roll Mode is turned on after Average has been set, Average is disabled and Normal sampling is enabled. If Average is selected while Roll Mode is turned on, Roll Mode is forcibly disabled.

High Resolution

In High Resolution mode, when the sampling rate is set lower than the optimum sampling rate, this mode averages and displays data captured by the optimum sampling. Since it is able to dampen random noise and effectively raise the vertical axis resolution, it becomes possible to smoothly monitor waveform traces. High Resolution mode is effective for both single-shot signals and repeated signals. The number of bits added for vertical axis resolution depends on the sampling rate (time/div setting, i.e., sweep speed). The lower the sampling rate (time/div setting) becomes, the greater the number of samples that are averaged for each display point. For example, the vertical axis resolution is increased by 1 bit each time the number of sampling times is quadrupled. The relation between the sampling rate and the vertical axis resolution (effective bit count) is shown in Table 9.

Table 9: Sampling Rate and Vertical Axis Resolution

Sampling rate (S/s)		Vertical Axis Resolution
Non-interleave: Up to 1 GS/s	**Interleave: Up to 2 GS/s**	**(bits)**
250 MS/s < sampling rate ≦ 1GS/s	500 MS/s < sampling rate ≦ 2 GS/s	8
50 MS/s < sampling rate ≦ 250 MS/s	100 MS/s < sampling rate ≦ 500 MS/s	9
10 MS/s < sampling rate ≦ 50 MS/s	20 MS/s < sampling rate ≦ 100 MS/s	10
2.5 MS/s < sampling rate ≦ 10 MS/s	5 MS/s < sampling rate ≦ 20 MS/s	11
Sampling rate ≦ 2.5 MS/s	Sampling rate ≦ 5 MS/s	12

Equivalent Sampling (Equ)

When the signal to be observed is a repetitive signal, Equivalent Sampling can be used to raise the apparent sampling rate for measurements. Using the trigger time T as a basis, data acquired after times t1, t2, and t3 are indicated by △, ○, and × respectively. This tn (n = 1, 2, 3…) time is measured and compiled with consideration given to relative time relationships so that a very high sampling rate can be realized.

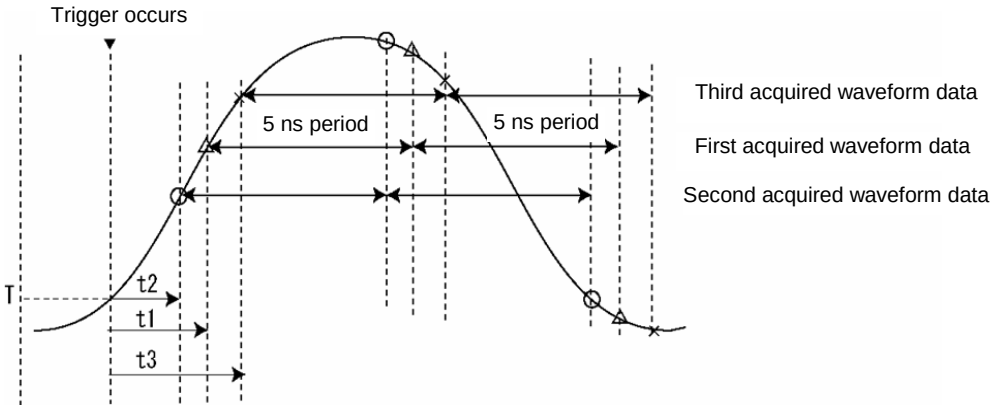


Figure 15: Equivalent sampling

The tn time measurement uses a resolution of 10 ps to realize the equivalent of 100 GS/s sampling.

NOTE: When Equivalent Sampling is active Max Memory Length will be ignored and set to 10 k.

To turn on Equivalent Sampling:

1. Press the Horizontal **SETUP button** to open the Horizontal menu.
2. Touch **Equivalent Sample** until **On** is selected.

Roll Mode

Roll mode displays, in real time, incoming points in single-shot acquisitions that have a sufficiently low data rate. This mode is automatically invoked for slow acquisitions (see Table 11).

NOTE: Roll Mode is not a sampling/acquisition mode, but a display mode that may be added when using certain acquisition modes.

Roll Operation

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.

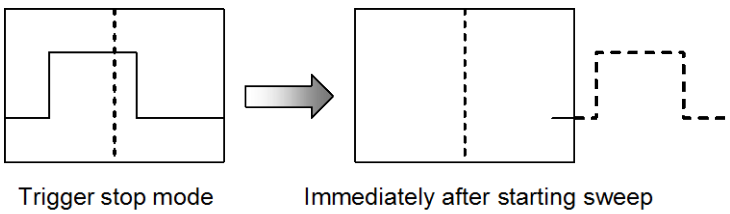


Figure 16: Roll mode operation

Roll Mode operates differently depending on the Trigger Mode:

- **AUTO Mode**, Roll Mode continues real-time acquisition and display
- **NORMAL Mode**, Roll Mode repeats a single-shot acquisition
- **SINGLE Mode**, Roll Mode acquires one trigger signal and then stops

Roll Mode only operates at the following timebases:

Table 10: Timebase and Memory Settings for Roll Mode Operation

Max Memory	Data Count per div	Time/div for Roll Operations
500	50	50 ms/div to 50 s/div
1 k	100	
5 k	500	
10 k	1, 000	
50 k	5, 000	
100 k	10, 000	500 ms/div to 50 s/div
500 k	50, 000	
1 M	100, 000	1 s/div to 50 s/div
2.5 M	250, 000	5 s/div to 50 s/div
5 M	500, 000	5 s/div to 50 s/div

Relation to Other Functions

- **Average Acquisition Mode**. If Average acquisition mode is selected while Roll Mode is on, Roll Mode is automatically turned off. The message “Roll Mode Off” appears in the Message bar.
- **Zoom**. Zooms can only be displayed once the Roll Mode acquisition has completed. Therefore, when the oscilloscope is in Auto Mode a zoom trace cannot be displayed while Roll Mode is enabled.

Horizontal Preference

The Horizontal Preference setting determines how the oscilloscope allocates memory.

- When Replay is the preference (default), the maximum number of sweeps are available for Replay. This preference can limit the maximum amount of adjustable skew in some timebase settings.
- When Deskew is the preference, the adjustable range of the skew is 0-500ns at all timebases. The number of sweeps available for replay decreases compared to what is available when Replay is the preference.

Table 11: Deskew Range When Horizontal Preference Is Replay

Time/div	Max Memory Length 5k to 5M points	Max Memory Length 1k points	Max Memory Length 500 points
50 ns/div or less	0 to 5.00 ns	0 to 5.00 ns	0 to 5.00 ns
100 ns/div or more	0 to 10.00 ns	0 to 10.00 ns	
200 ns/div or more	0 to 20.00 ns		
500 ns/div or more	0 to 500 ns		

To change the Horizontal Preference setting:

1. Press the Horizontal **SETUP button**. The Horizontal menu opens.
2. Touch **Preference** until the desired setting is selected.

Max Memory Length

The Maximum memory length will determine the size of the waveforms captured during each acquisition. The max memory setting can have impact on other oscilloscope functions such as Roll Mode, Replay, and FFT operation.

To set Max(imum) Memory Length:

1. Press the Horizontal **SETUP button**. The Horizontal menu opens.
2. Touch **Max Memory Length** until the desired setting is selected.

Half vs. Full Channel Mode

When no more than one channel each from the (CH1, CH2) pair and the (CH3, CH4) pair is displayed, the oscilloscope is said to be in half-channel mode.

When operating in half-channel mode, the oscilloscope may be set to the maximum sampling rate at 2 GS/s (with the interleave function) and double the memory length (with the channel combining function).

Interleave function

When in half channel mode, interleave operation is performed to change the 1 GS/s sampling rate to 2 GS/s.

NOTE: Operation at 2 GS/s also depends on the Max Memory Length setting and the sweep range (time/div) setting.

Channel combination function

When in half channel mode, channel combining is performed to obtain the max memory length.

Zoom

The Zoom function creates a separate, enlarged display of a specified part of a waveform while retaining the display of the original acquired waveform.

Zoom Set Up

1. Turn on channels and adjust Vertical and Horizontal scale of the source traces as desired.
2. Press the front panel **Zoom button**. The button lights to show you are now in Zoom mode.
3. Turn the **Time/div knob** to increase/decrease the zoom scale. Press the knob to return to the original scale.
4. Press the Zoom button again to exit Zoom mode.

Zoom Display

Main Window/Zoom Window

When Zoom is turned on, the display splits into two, 8 x 10 grid areas. The Main window on top shows the source traces, and the Zoom window below it shows the zoom traces. The zoom trace timebase (Time/div), indicated by the  icon, is displayed in the Acquisition Status bar to the left of the channel trace timebase. For repetitive waveforms (such as Auto trigger), this is initially the same scale as the source trace.

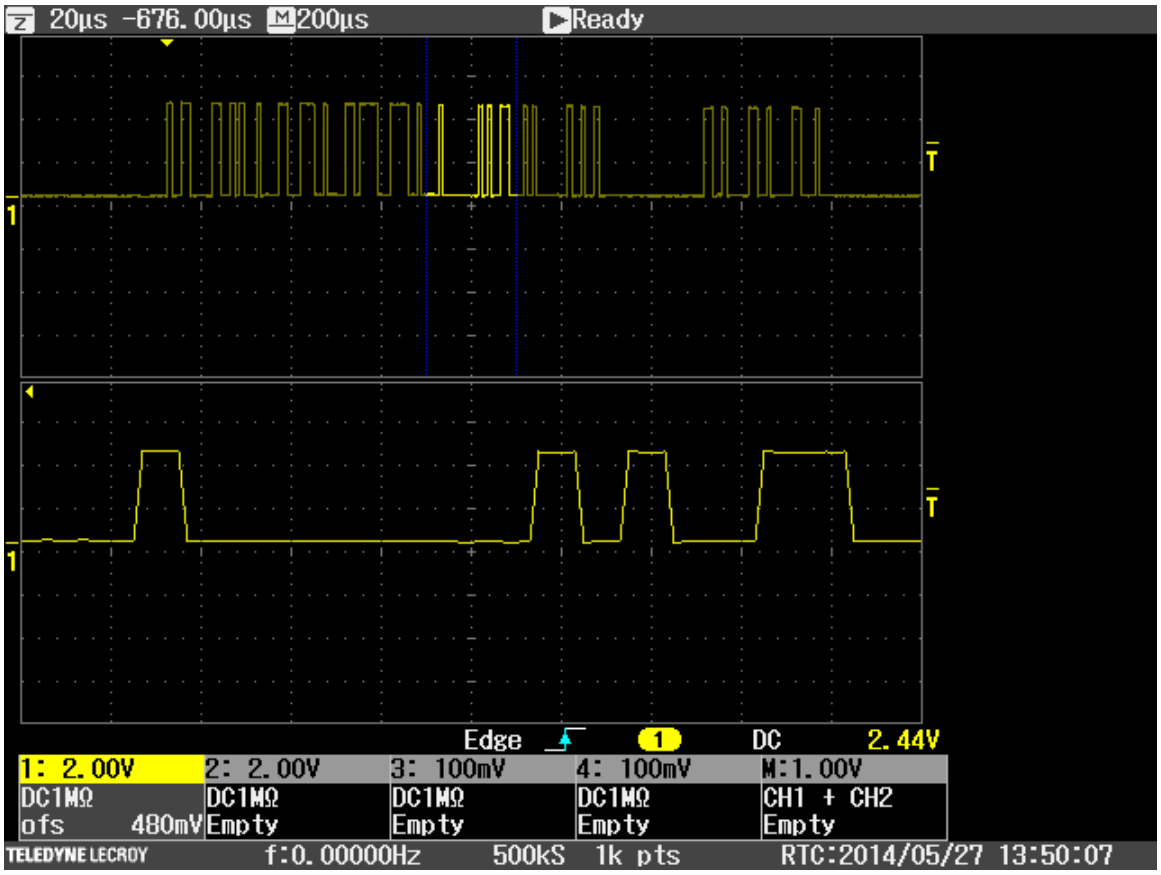


Figure 17: Zoom display

All active (on) traces are zoomed together. Just as all channel traces share the same timebase, so do all zooms share the same timebase, which is likely to differ from the channel timebase.

Range Setting Cursors

Two range setting cursors appear as vertical blue lines at the outside edges of the Horizontal axis in the Main window. These mark the Horizontal range of the source trace that is being viewed in the full width of the Zoom window. The area between the cursors is highlighted to show the portion of the source trace displayed in the Zoom window, while the area outside them is dimmed to indicate it is not currently shown.

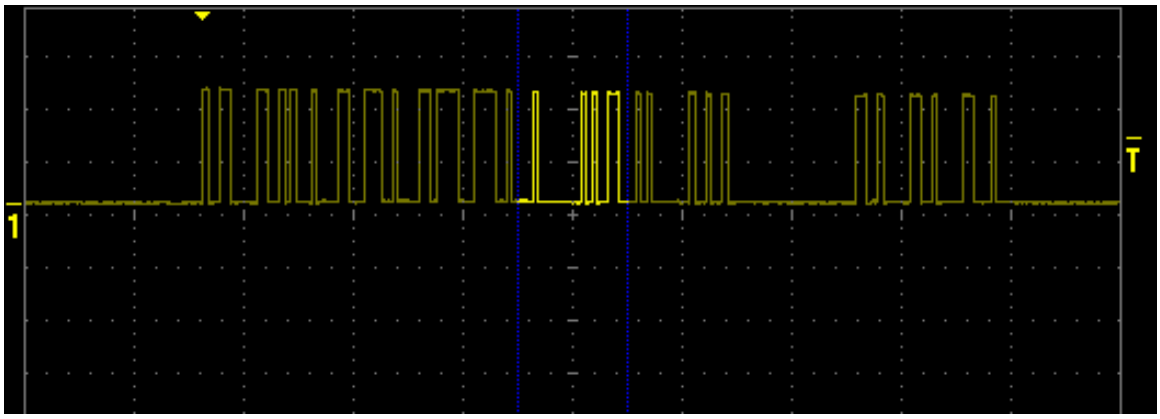


Figure 18: Zoom range setting cursors

Turn the Time/div knob to move the range setting cursors. By narrowing/widening the span of the cursors, you increase/decrease the zoom scale. Press the Time/div knob to return to the full range (original scale).

NOTE: If you exit and re-enter Zoom mode without changing any other acquisition settings, the zoom opens using the last range set. If you exit Zoom mode while the oscilloscope is stopped the zoomed timebase will be displayed.

When Zoom is on, the Horizontal DELAY knob changes the zoom trigger delay setting only, not the channel trigger delay. As the zoom delay changes, the range setting cursors will track together across the source trace, highlighting that portion that is now displayed in the Zoom window.

Replay

All acquisitions are saved in the oscilloscope buffer until full. Replay mode is always running so there is no need to turn it on. When the trigger is stopped, you can “replay” these previous acquisitions, allowing you to closely observe changes in the signal over time.

1. Press the **RUN/STOP button** to stop the acquisition. The Replay icon is automatically lit. The current/total number of sweeps is shown on the Acquisition Status bar at the top of the screen.
2. Turn the **Replay knob** to go back or forward through the sweeps.
3. Press the **Replay knob** to jump to the most recently acquired waveform.
4. Press **RUN/STOP** again to exit Replay mode and resume acquisition.

NOTE: Replay data only persists until the buffer is full or the Vertical and Horizontal setups change. Then they are overwritten by new data.

The total number of sweeps that are potentially available for Replay depends on the Max Memory Length setting, as well as the Horizontal Preference setting. A Horizontal Preference of Replay will optimize the number of sweeps.

Table 12: Max Memory Length vs. Number of Sweeps

Max Memory (points)	Total Number of Sweeps
500	2048
1k	2048
5k	1024*
10k	512*
50k	128*
100k	64*
500k	16*
1M	8*
2.5M	2*
5M	1*

* When in Half Channel mode (one channel each from CH1-CH2 and CH3-CH4 pairs).

Relation to Other Functions

- **Max Memory.** If the memory setting yields a total of 1 sweep, the Replay function cannot operate.
- **Average Mode.** Replay does not work on waveforms acquired using Average sampling mode.
- **Equivalent Sampling (Equ).** Replay does not work on waveforms acquired using Equivalent Sampling.

Cursors

Cursors are makers 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 two principal types of cursors available in the WaveJet Touch.

- **Time** and **Amplitude** cursors show the values where the cursor markers intersect with the waveform on the Y or X axes. They also show the Delta time/amplitude represented by the difference between the first and second cursor markers, and the waveform frequency calculated from the measured Delta Time. There is an option to apply the Time and Amplitude cursors together, maximizing the readout options.
- **Value At** cursors show the amplitude values where each cursor intersects the waveforms.

Turn on Cursors

1. Press the front panel CURSORS button. The Cursors menu appears showing the current cursor selection in the top cell.
2. Continue to press the CURSORS button until the desired type of cursor is selected or touch **Type** and chose the desired cursor.
3. Touch **Source** and choose the source channel to measure. You may also choose Math to measure on the Math trace.

Adjust Cursor Position

When cursors are turned on, cursor marker lines appear over the display. The boldest line is the active one that can be adjusted.

1. Push the ADJUST knob to select the line you wish to reposition.
2. Turn the ADJUST knob until it is at the desired position. You will see the cursor readouts change as you move the line.
3. Repeat until all the cursors are repositioned.

Cursor Readout

Trigger Status Bar

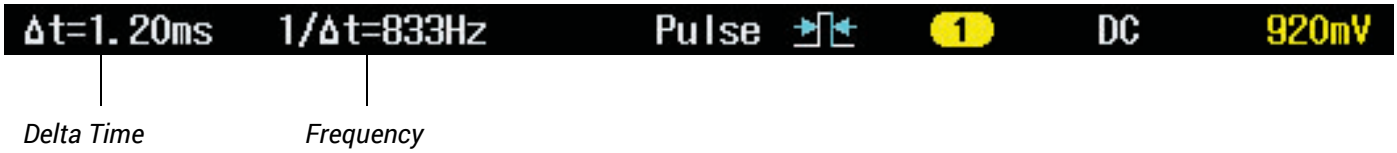


Table 13: Trigger Status Bar Cursor Readouts

Readout	Screen Display	Meaning
Delta Time	Δt = ± *nnn uu*	For Time cursors, delta time between the two cursor markers.
Frequency	1/Δt = ± *nnn* uHz	Frequency (in Hz) calculated from Δt measurement
	Δf	Frequency calculated for FFT waveform.

Channel and Math Descriptor Box

Figure 19: Cursor readouts on channel and math descriptor boxes

Table 14: Descriptor Box Cursor Readouts

Readout	Screen Display	Meaning
Delta Voltage	ΔV *nnn uu*	For Amplitude cursors, voltage represented as the difference between cursor markers. If the units are other than V, this readout will still begin with ΔV, but the actual unit will be shown after the value. In the case of Derivative and Integral, the ΔV reading on the Math Descriptor Box represents the voltage after the calculation has been made. In the case of XY traces, an x or y is shown after ΔV.
Delta Decibels	ΔdBm *nnn* dBm	For FFT, decibels represented as the difference between cursor markers, shown on Math Descriptor Box. The Delta Voltage of the FFT source channel is also shown on the Channel Descriptor Box.
Value At	V@ *nnn uuu*	For Value At Cursor, the measured value where the cursor marker intersects with the source waveform. The first cursor marker is represented by a dotted line, the second by a dashed line.

Cursor Menu

Table 15: Cursor Menu Cursor Readouts

Readout	Screen Display	Meaning
Measured Time		This is the actual value where the cursor marker intersects with the waveform. It is always shown in the current timebase. The first cursor marker is represented by a dotted line, the second by a dashed line.
Measured Amplitude		Same as Time, except shown in the Vertical Units of the source channel. For Value At Cursors, this readout is shown on the Channel Descriptor Box instead of the Cursors menu.

Relation to Other Functions

- **Zoom.** When the Zoom function is on, the zoom trace in the bottom window is the target for cursor measurements, although the lines will appear to intersect with all active traces.
- **Persistence**. When Persistence display is turned on, the Value at Cursor measurement is performed for the most recent waveform acquired.
- **Roll Mode**. When Roll Mode is turned on, the Value at Cursor measurement that normally appears in the Channel Descriptor Box is "******".

Measure

The WaveJet Touch measurement functions include:

- **Auto Measure**— all active measurement parameters will measure all occurrences of a parameter in a single acquisition, with the option to display statistics (Min, Max, Num) as well as the first measured value following the trigger.
- **Pass/Fail Testing**
- **Logging**

Auto Measure

Measurements may be applied to any active channel or to the Math trace.

Set Up Measurements

1. Press the font panel **MEASURE button**.
2. Touch **Measure No.** and choose one of the measurement slots, A –D. Alternatively the **Adjust** knob can be used to change the measurement slot.
3. Touch the **Source** and choose the signal to measure, CH1-CH4, or the calculated Math trace.
4. Touch **Measure Item** and choose one of the measurement types, then select the measurement. See Table 16 for a list of measurements by type.

   NOTE: You may need to navigate through several menus to find your choice. If necessary, press CLOSE MENU to return to the previous menu.
5. Repeat Steps 3 to 5 until you have completed your measurement setup.
6. To display the measurements now, touch **Next**, then touch **Measure** until **On** is selected.

   NOTE: You can preset measurements but turn Off the measurement display to maximize waveform space on the grid. The measurement setup remains for you to turn On when ready to make the measurements.
7. Touch **Min/Max** and choose the calculation method:
   - When Min/Max is Off, Auto Measure is performed asynchronously in relation to waveform acquisition.
   - When Min/Max is On, Auto Measure is performed in sync with waveform acquisition, and the statistical value are added to the measure readout.

Measurement Readout

When measurements are turned on, a readout table appears below the grid.

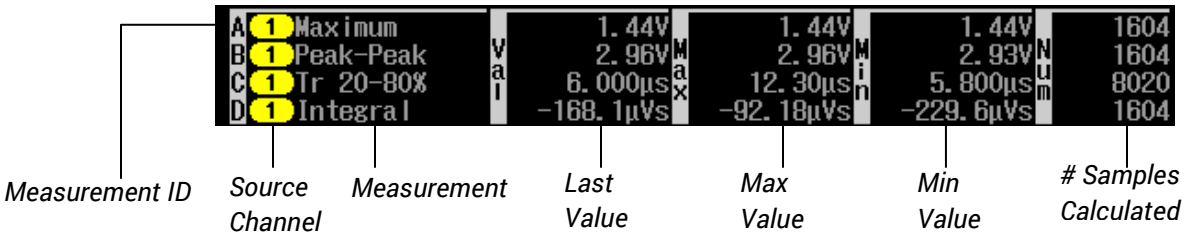


Figure 20: Measurement table

If Min/Max (statistics) is set to Off, the last three columns of the table are left blank, and only the first measured value following the trigger is displayed.

Available Measurements

Table 16: Measurements

Type	Measure Item	Summary	Min/Max Off (Val only)		Min/Max On		Unit
			Target[1]	Freq.	Target[1]	Freq.	
Vertical	Maximum	Maximum value	Entire waveform	1	Entire waveform	1	V
	Minimum	Minimum value	Entire waveform	1	Entire waveform	1	
	Peak-Peak	Differential between maximum value and minimum value	Entire waveform	1	Entire waveform	1	
	RMS	Root mean square	Entire waveform	1	Entire waveform	1	
	Cycle RMS	Cycle root mean square	Integer cycle (N cycle)	1	Integer cycle (N cycle)	1	
	Mean	Mean	Entire waveform	1	Entire waveform	1	
	Cycle Mean	Cycle mean	Integer cycle (N cycle)	1	Integer cycle (N cycle)	1	
	Top [2]	Top value	Entire waveform	1	Entire waveform	1	
	Base [2]	Base value	Entire waveform	1	Entire waveform	1	
	Top-Base[2]	Differential between top and base values	Entire waveform	1	Entire waveform	1	
	+OverShoot	Positive overshoot	Part first detected	1	Part first detected	1	
	–OverShoot	Negative overshoot	Part first detected	1	Part first detected	1	
Horizontal	Tr20-80%[2]	20% to 80% of rise time	First rising edge	1	Rising edges	N[3]	s
	Tf80-20%[2]	80% to 20% of fall time	First falling edge	1	Falling edges	N[3]	
	Tr10-90%[2]	10% to 90% of rise time	First rising edge	1	Rising edges	N[3]	
	Tf90-10%[2]	90% to 10% of fall time	First falling edge	1	Falling edges	N[3]	
	Frequency	Frequency[4]	First cycle	1	Integer cycle (N cycle)	N[3]	Hz
	Period	Period[4]	First cycle	1	Integer cycle (N cycle)	N[3]	s
	No of +Pulse	Positive pulse count[4]	Entire waveform	1	Entire waveform	1	pulse
	No of –Pulse	Negative pulse count[4]	Entire waveform	1	Entire waveform	1	
	+PulseWidth	Positive pulse width[4]	First positive pulse	1	Positive pulse	N[3]	s
	–PulseWidth	Negative pulse width[4]	First negative pulse	1	Negative pulse	N[3]	
	Duty Cycle	Duty cycle[5]	First full cycle	1	Integer cycle (N cycle)	N[3]	%

Type	Measure Item	Summary	Min/Max Off (Val only)		Min/Max On		Unit
			Target[1]	Freq.	Target[1]	Freq.	
Other	Integral	Integral	Entire waveform	1	Entire waveform	1	Vs
	Skew	Time differential between measurement points. Specify this relatively, at a rate based on amplitude. Each source has the parameters: • Level1, Level2 • Peak-to-Peak % • Slope1, Slope2 • Rise/fall	Pair first detected	1	Pair first detected	1	s
	Skew@Level	Time differential between measurement points. Specify level as an absolute value. Each source has the parameters: • Level1, Level2 (at full scale of setup range) • Peak-to-Peak % • Slope1, Slope2 • Rise/fall	Pair first detected	1	Pair first detected	1	

1) Default target is the entire waveform. When Zoom or Time Cursors are applied, target is the section of waveform that appears between markers. Changing the placement of these will affect the measurement.
2) When either a valid top value or a valid base value are measured, the measurement result becomes an invalid display "*********".
3) Because the cycle and frequency differ between waveforms, this is assumed N.
4) Peak-to-Peak measures the difference between the waveform's maximum value and its minimum value within the measurement range. For these measurements the level is fixed at 50%.
5) Duty Cycle measures the ratio of the waveform's cycle to its pulse width within the measurement range.

No. of Pulses (Pulse Count)

This measures the waveform's pulse count within the measurement range.

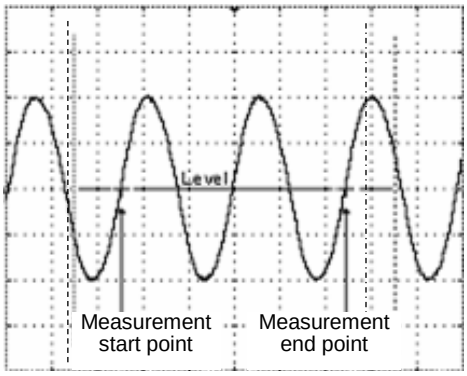
- **No of +pulses**—One pulse is counted when the waveform passes from the base to the top and then from the top back to the base, based on a level that is 50% of Peak-to-Peak.
- **No of –pulses**—One pulse is counted when the waveform passes from the top to the base and then from the base back to the top, based on a level that is 50% of Peak-to-Peak.

CYCLE RMS AND CYCLE MEAN

This measures the rms of the integer cycle part of the waveform in the measurement range.

- **Cycle RMS**—The period during which a level based on 50% of the Peak-to-Peak level is passed within the measurement range is determined and the rms of that period is measured.
- **Cycle Mean**—The period during which a level based on 50% of the Peak-to-Peak level is passed within the measurement range is determined and the mean of that period is measured.

In the following figure, the rms and mean are measured from the measurement start point where the level was fixed as 50% of Peak-to-Peak to the measurement end point.



Cycle RMS: 72.00 mV

Cycle Mean: 1.00 mV

50 mV/div

200 ns/div

Figure 21: Cycle RMS and cycle mean

OVERSHOOT

This measures overshoot in rising and falling signals within the measurement range.

- **+Overshoot**—The rise that occurs where 50% of Peak-to-Peak within the measurement range crosses the first waveform data is detected, and overshoot in that rise is measured. It is calculated as:

  +Overshoot value (%) = 100 × [(+ peak value) – top value] /
  (Top value – base value)

- **–Overshoot**—The fall that occurs where 50% of Peak-to-Peak within the measurement range crosses the first waveform data is detected, and overshoot in that fall is measured. It is calculated as:

  –Overshoot value (%) = 100 × [base value –(– peak value] /
  (Top value – base value)

INTEGRAL

This measures the area (integral) of the waveform in the measurement range. In area (integral) measurements, measurements are based on the reference level (GND level) of the channel to be measured (set via the vertical axis offset control knob).

In the example below, the reference level (GND level) is based on the measurement range sandwiched between two time points, and integral values are measured at S1+S2+S3+S4+S5+S6 (sections filled in black).

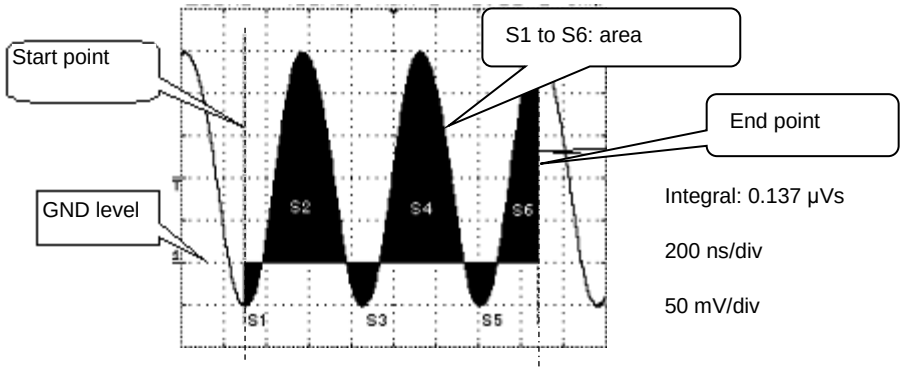
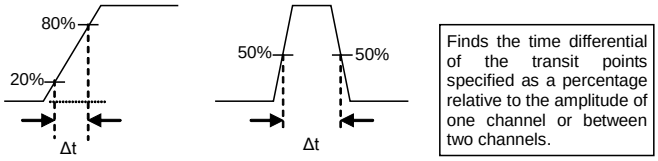


Figure 22: Integral measurement

SKEW / SKEW@LEVEL

This measures the time differential (skew) between the selected waveform's rising (or falling) edge1 and its falling (or rising) edge2. Edge 1 and edge 2 can be set either between two CH selected or in CH the same.

- **Skew**—Specifies the measurement point's level as a percentage relative to the amplitude. This measurement is illustrated in the figure below.



- **Skew@Level**—Specify the measurement point's level as an absolute value. This measurement is illustrated in the figure below.

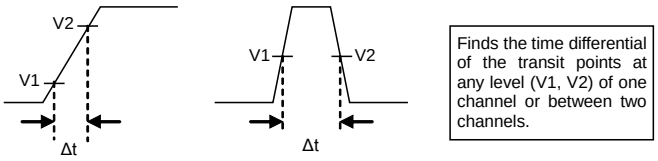


Figure 23: Skew and Skew@level measurements

NOTE: Edge 2 is measured at the same time when edge 2 is detected. Therefore, edge 2 needs to occur before edge 1 otherwise the measurement could display "*********" (invalid measurement).

Using Auto Measure with Zoom or Cursors

When Zoom is turned on, measurements apply to the zoom trace rather than the channel trace. The Zoom range setting cursors (blue vertical lines that intersect with the time axis) will act as measure gates marking the section of the source waveform that is measured—in other words, the zoom.

Similarly, when Time or Value at cursors are turned on, measurements will only apply to the section of the source waveform between the white, vertical cursor lines.

Relation to Other Functions

- **Inversion.** When measuring inverted waveforms, values for sign, slope, etc. are displayed in reverse type. Also, Tr, Tf, Top, and Base are themselves handled as inverted, as are the measured values for RMS, Mean, Maximum, and Minimum.
- **Math.** The following apply when measuring a Math waveform resulting from the multiplication, integral or derivative operations:
    - Vertical Measure Items Top, Base, Top-Base, +Overshoot and –Overshoot cannot be measured. When any of these five Measure Items is selected, only "*********" is shown on the measurement readout table.
    - Horizontal Measure Items Tr and Tf are calculated from Peak-to-Peak rather than Top-Base.
- **Clear.** During Run mode (Auto, Normal, or Single acquisition is on), measurement statistics (Min, Max, Num), Pass/Fail judgment results, and Log entries are reinitialized by pushing the CLEAR button.
- **Roll Mode.** Auto Measure does not work in Roll mode. All measurement displays become "*********".

Pass/Fail Testing

Pass/Fail judgments can be made on any waveform measurements compared to the user-defined test conditions, or to the waveform samples compared to a mask overlay (mask testing).

Pass/Fail Readout

Pass/Fail judgments for individual measurements appear on the Measurement Table. The value shown in the Val column will be green if Pass or red if Fail. The overall Pass/Fail judgment and test counter appears in the Message bar, replacing the frequency counter when Pass/Fail testing is on. Mask test results appear only in the Message bar.

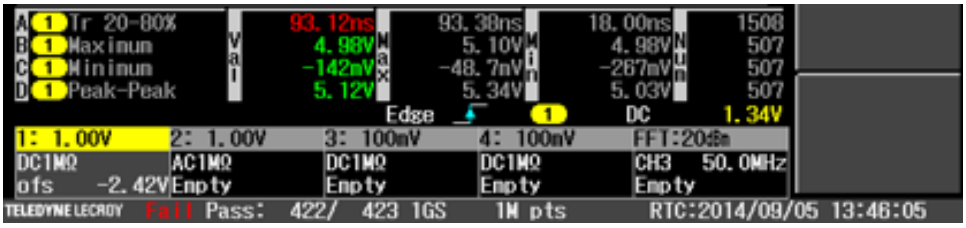


Figure 24: Measurement table with Pass/Fail test results

Table 17: Pass/Fail Readout in Message Bar

Item	Format	Comments
Pass/Fail judgment result	Pass: or Fail:	Replaces frequency counter display when P/F testing is turned On.
Counts	nnnnn/nnnnn	Pass counts / Total judgment counts

Pass/Fail Measurement Testing

Measurement testing takes a set of measurements made on one or more target waveforms and compares each measurement to a user-defined condition. If a single measurement in the set fails to meet its test condition, the entire test Fails. Actions can be set on either a Pass or Fail result.

NOTE: As with any measurements, the target for the measurement is the area between cursor markers when Time cursors are applied or the zoomed trace when zoom is active.

Set Up Measurements

Before setting up the Pass/Fail test conditions, configure measurements A–D. See Set Up Measurements.

Set Up Conditions

1. Press the **MEASURE button** to open the Measure menu, then turn **Measure On**. This is required to view the Pass/Fail results. It is helpful to also turn Min/Max On.
2. Touch **Pass/Fail**, then turn **Operation On**.
3. Touch **Judge By** until **Measure** is selected.
4. Touch **Condition**.
5. Touch **Pass If** and choose the formula that expresses the Pass condition for the measurement shown in Source. To exclude this measurement from the test, choose Any Value.

   NOTE: In all formulas, Value stands for the measured value (what appears in the Val column on the readout). The letters m and n represent user-defined variables.
6. Define the values for **m** and **n**:
   - Touch **m/n** until the top field is selected, then **turn ADJUST** to set the value.
   - To also change units, **push ADJUST** to go to the bottom field, then turn it to select the unit.
7. Touch **Source** to change the measurement, then repeat Steps 5-6.

   NOTE: All active measurements should have a Pass If condition set or be excluded by using Any Value. At least one measurement must have a condition.

Set Up Actions

These steps are optional:

1. Press **CLOSE MENU** to return to the Pass/Fail menu.
2. Touch **Actions**.
3. Touch each action on the list and choose for it to occur when there is a Pass or a Fail, or:
   - To stop an action from occurring regardless of the test result, choose None.
   - To invoke the action on every trigger, whatever the Pass/Fail result, choose Any Result.

See Actions for a description of each action.

USING PAGE SEARCH

Page Search allows you to review Pass or Fail judgments from the measurement history saved in the log file.

1. Press **RUN/STOP** to stop acquisition.
2. Press the **MEASURE button**, then touch **Pass/Fail > Page Search**.
3. Touch **Search For** and choose to view Pass or Fail results.
4. Touch **Previous** to go to the previous instance of that judgment (from your current place in the history), or **Next** to go to the next.

Pass/Fail Mask Testing

SET UP CONDITIONS

1. Press the **MEASURE button** to open the Measure menu.
2. Touch **Pass/Fail**, then turn **Operation On**.
3. Touch **Judge By** until **Mask** is selected.
4. Touch **Condition**.
5. To view the mask over the waveform, turn **Display On**.

   NOTE: The mask used for the test is whatever was last created on or recalled to the oscilloscope. Use the create or recall procedures to change the mask.
6. Touch **Source** and choose channel to test against the mask.
7. Touch **Pass If** and choose for samples to be All In or All Out of the mask area to constitute a Pass.

CREATE/EDIT MASK

To create a new mask from a live signal, or to change the size of the mask:

1. On the Mask Condition menu, touch **Edit Mask**
2. Touch **Source** and choose the waveform from which to build the mask. This may be a live channel trace, the Math trace, or one of the saved Reference Waveforms (REF1-REF5). Leave as is when editing the current mask.
3. Touch **Horizontal**, then turn **ADJUST** to enter the portion of a Horizontal division around the waveform for the mask to fill.
4. Touch **Vertical** and repeat for the portion of a Vertical division.
5. Touch **Make Mask**.

TIP: To revert to the previous mask display, touch **Undo Mask**.

Save Mask File

To save the current mask to USB memory for later recall:

1. Insert the drive into the front panel USB port.
2. On the Mask Condition menu, touch **Save/Recall Mask**
3. Touch **Save/Recall** until **Save** is selected.
4. Optionally, touch **File Name** and enter a new name for this mask file:
   - With the first position selected, **turn ADJUST** to select a character from the Character List popup.
   - **Push ADJUST** to move to the next position and repeat.

NOTE: The default filename is MASK0000. The four digits at the end increment each time a new mask is saved.

Recall Mask File

To recall an existing mask from USB memory:

1. Insert the drive into the front panel USB port.
2. On the Mask Condition menu, touch **Save/Recall Mask.**
3. Touch **Save/Recall** until **Recall** is selected.
4. Touch **File List**, then **turn ADJUST** until the desired mask file is highlighted. **Push ADJUST** to select it. Ensure that the Rotate setting is set to **Select File**

   NOTE: Up to 20 files may be displayed at once. When there are more than 20 files, continue to turn ADJUST to scroll to the next page or press ADJUST so that Select Page is the current Rotate setting.

Set Up Actions

These steps are optional:

1. Press **CLOSE MENU** to return to the Pass/Fail menu.
2. Touch **Actions** to open the sub-menu.
3. Touch each action on the list and choose for it to occur when there is a Pass or a Fail, or:
   - To stop an action from occurring regardless of the test result, choose None.
   - To invoke the action on every trigger, whatever the Pass/Fail result, choose Any Result.

Pass/Fail Actions

These optional actions can be taken depending on a Pass or Fail result. The actions are the same for measurement and mask tests.

Stop Sweep

Stops the acquisition at the sweep that Fails or Passes, depending on the result selected.

Save Screen

Captures the entire display (waveform plus mask or measurements) and saves it as an image file according to your current print to USB Memory settings.

When taking this action, make your file format and file name selections prior to starting the Pass/Fail test. See Print to File. Insert a memory drive into the front panel USB port; the image will be copied onto this.

NOTE: When Save Screen is set as a Pass/Fail action, the following functions are disabled for as long as P/F testing is enabled: Save/Delete to USB Memory, Recall from USB Memory, Print button, Save/Recall Mask, (Firmware) Update (in Status & Update), USB Memory Information (in Status & Update).

Save Waveform

Captures waveform data, panel setups, and other information and saves it as a waveform file according to your current Save/Recall settings.

When taking this action, make your file format and file name selections prior to starting the Pass/Fail test. See Save to USB Memory. Insert a memory drive into the front panel USB port; the image will be copied onto this.

NOTE: When Save Waveform is set as a Pass/Fail action, the following functions are disabled for as long as P/F testing is enabled: Save/Delete to USB Memory, Recall from USB Memory, Print button, Save/Recall Mask, (Firmware) Update (in Status & Update), USB Memory Information (in Status & Update).

Output Pulse

Sends a pulse from the back panel Aux Out connector. The output pulse has an amplitude greater than 3 V into 1 MΩ and 0.9 V into 50 Ω. When the output pulse is set to a pass/fail test result the pulse width is around 8-10 ms.

Beep

Emits a beep.

Measurement Logging

The logging function records measurements and Pass/Fail judgment results (along with timestamps) to internal memory. Up to 86,400 log entries can be recorded. After this number, new entries will overwrite the old. Log files can be displayed on screen or saved to USB memory.

Set Up Logging

1. Press the front panel **MEASURE button**, then touch **Logging**.
2. Turn **Capture On**.

Display Logs

This function displays the most recent 16 entries from the logs recorded in internal memory in a pop-up window.

1. Press **RUN/STOP** to stop the acquisition.
2. Press the **MEASURE** button, then touch **Logging**.
3. Touch **Show Log Entries**.
4. Turn the **ADJUST knob** to scroll the list of entries. Ensure that the Rotate setting is set to **Select File**.

   NOTE: Up to 20 entries can be displayed at once. Continue turning ADJUST to scroll to the next page of entries or press ADJUST so that Select Page is the current Rotate setting.
5. Touch the **CLOSE MENU** button to close the log display.

Save Log File to USB Memory

To export the entire contents of the internal log to a log file:

1. Insert a drive into the front panel USB port.
2. Press the **MEASURE** button, then touch **Logging**.
3. Optionally, touch **File Name** and enter a new name for the log file:
   - With the first position selected, **turn ADJUST** to select the first character from the Character List.
   - **Press ADJUST** to move to then next position and repeat.

   NOTE: The default name is LOGM0000. The four digits at the end increment each time a new log file is saved.

Math

A Math trace shows the result of a mathematical operation applied to one or more source channel traces. Cursors and Measurements can be applied to Math traces the same as for Channel traces.

Available Math operations include:

- Add, Subtract, Multiply, Divide
- Integral
- Derivative
- Fast Fourier Transform (FFT)

NOTE: The Integral, Derivative, and FFT operations can also be applied to the result of the basic Math operations (e.g., Add, Subtract, Multiply, Divide).

Turn On Math Trace

Press the front panel MATH button to enable the Math trace and open the Math setup menu. The Math descriptor box changes from grey to the red to indicate it is now on and a red Math trace appears on the grid using the last saved Math setups. The Math setup menu can also be accessed by touching the Math descriptor box.

Math Readout

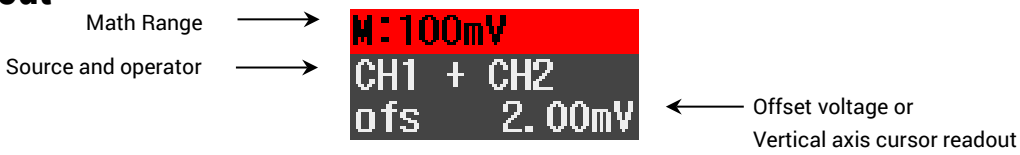


Figure 25: Math descriptor box when trace is on

Table 18: Math Descriptor Box Elements

Setting	Screen Display	Meaning
Math Range	M:*nnn u*V M:*nnn u*A M:*nnn u*W M:*nnn u*C M:*nnn*dBm	M: followed by the math trace Vertical range. During FFT operation, the Vertical range is shown in decibels (per division).
Math Operation	CH*n1* + CH*n2*	Addition of two channels shown
	CH*n1* - CH*n2*	Subtraction of channel *n2* from *n1*
	CH*n1* * CH*n2*	Multiplication of two channels shown
	*n*FFT FFT（CH*n1* <operator> CH*n2*）	FFT of channel *n* or FFT taken after an arithmetical operation.
	∫(CH*n*)dt	Integral of channel *n*
	d(CH*n*)/dt	Derivative of channel *n*
Math Offset	ofs *nnn u*V ofs:*nnn u*A ofs *nnn u*W ofs:*nnn u*C pos *n.nn*dBm	Math trace Vertical offset position. During FFT operation, the position is shown in decibels. This display is replaced by the cursor readout if cursors have been applied to the math trace. See Cursors.

The units that appear on the Math descriptor box readout will reflect any Rescaling that has been applied to the Vertical axis range of a source channel.

Table 19. Math Units After Rescaling

Operation	CH*n1*	CH*n2*	Math Unit
× operation	V	V	VV
	A	V	VA
	A	A	AA
+, – operations	V	V	V
		A→V (converted to voltage)	V
	A	A	A

Addition, Subtraction, Multiplication

1. Input the source signals. The waveforms should appear on screen.
2. Press the **MATH button** to display the Math menu.
3. Touch **Source** and select the first signal.
4. Touch **Operator**, then select the desired operator.
5. Touch **Source2** and select the second signal.

NOTE: The mathematical expression applied is exactly what is shown on the dialog and menu. In the case of subtraction, especially, the order in which you select the source signals can affect the result.

Integral

1. Input the source signals. The waveforms should appear on screen.
2. Press the **MATH button** to display the Math menu.
3. Touch **Source** and select the input signal.
4. Touch **Operator**, then **Next**, and choose **Integral**. An Integral waveform is displayed in red next to the source waveform from which it was calculated.

Derivative

TIP: When there is a lot of noise in the source signal for which you want to take the derivative, use Average acquisition mode and apply bandwidth filters to the source channel prior to setting up the Math trace.

1. Input the source signals. The waveforms should appear on screen.
2. Press the **MATH button** to display the Math menu.
3. Touch **Source** and select the input signal.
4. Touch **Operator**, then **Next**, and choose **Derivative**. The differentiated waveform is displayed in red next to the source waveform from which it was calculated.

FFT

With digital oscilloscopes, the observed signal data is normally shown as time axis area waveform data. Using the FFT operation, the observed signal data can be displayed as frequency area data, similar to a spectrum analyzer. One of the advantages of FFT over other frequency transformation methods is that it can be performed on both single-shot signals and slow repetitive signals. Also, high-speeds operations can be performed, similar to the oscilloscope's time domain recording function. Consequently, the FFT Math function can perform burst waveform analysis, frequency analysis or amplitude analysis of repetitive waveforms. Features include:

- **Power spectrum.** The energy (electric power) of each signal frequency component is shown in dBm units along the vertical axis. 1 dBm is the voltage equivalent to 1 mW when the load is 50 Ω (0.316 Vpk /0.224 Vrms).
- **Dedicated display.** Time axis (YT) waveforms and frequency axis (FFT) waveforms on are shown on the same screen. However, the YT and FFT waveform information is shown in separate waveform descriptor boxes, so that it is easily distinguished.
- **Cursor measurement.** Vertical axis components (dBm) and horizontal axis components (frequency in Hz) can be measured at the intersect point of the FFT waveform and the Vertical axis. Use the cursor type "Value at Cursor".

FFT Set Up

1. Input the FFT source signal. The waveform should appear on screen.
2. Press the **MATH button**. The Math menu appears in the right part of the screen.
3. Touch **Operator** and choose **FFT**.
4. On the Math menu, touch **FFT Window** and choose a window type. See the list below.

FFT Window

Three types of windows are provided so that different parameters can be emphasized when measuring.

- Rectangular, suited for analysis of transient phenomena in which a waveform continues beyond operation range, such as with a burst waveform.
- Von Hann, suited for analysis of repetitive waveforms.
- Flat Top, suited for analysis of amplitude in repetitive waveforms.

During an FFT operation, data within the operation range is handled as continuous. Consequently, if the input signal cycle is not kept within the operation range by the integer cycle, the time axis waveform will become discontinuous, causing the spectrum obtained by the FFT to spread. This is called leakage. Such spectral leakage can be prevented by using an FFT window that is suitable for the signal to be analyzed.

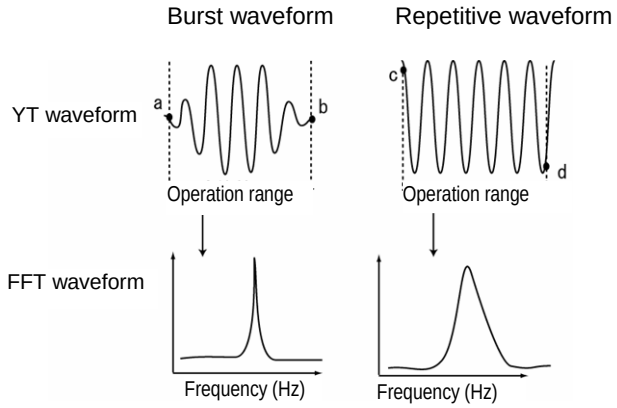


Figure 26: FFT operation range

In the burst signal shown above, the signal converges within the operation range and the signal is continuous between points a and b, in which case spectral leakage does not occur. In a repetitive waveform, however, points c and d before and after the operation range are discontinuous, so in the case of a repetitive signal, spectral leakage may occur and preclude accurate measurements. When using the FFT window, processing can be performed to reduce discontinuous points before and after the operation range (as shown in the FFT example above) which can suppress spectral leakage.

Target Data

The amount of target data in the FFT waveform display ranges from 4 to 8192.

Although Max Memory Length can be set within the horizontal axis menu, the amount of target data for FFT operations must be determined according to the Time/div and sampling rate (S/s). For example, if the number of points is set as 1 k (which is not 1024), the amount of target data for the operation is the number closest to a power-of-two number, which is 512. The FFT operation target is shown on the screen's left edge.

Aliasing

As a digital oscilloscope, the observable range of signal frequencies extends until one half the sampling rate frequency (Nyquist frequency). If components with a frequency higher than this are observed, they are observed as waveforms with frequency components that have lower frequency than the actual ones (this is called "aliasing").

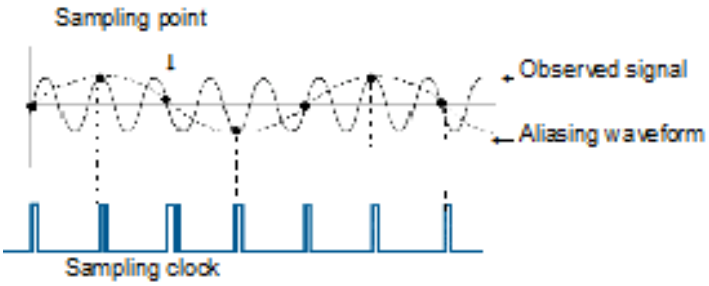


Figure 27: Aliasing

Similarly, when creating an FFT waveform, if the signal includes frequency components that have a higher frequency than one half the target data's sampling rate (Nyquist frequency), aliasing occurs, and the waveform appears to have frequency components lower than the actual ones. The sampling rate and frequency counter values are shown in the Message bar at the bottom of the screen. Be cautious when comparing these values to the observed waveform frequency.

Aliasing can be prevented in the following ways.

- Adjust in the direction of a higher sampling rate. Increase the Maximum Memory Length. While only the time axis (YT) waveform is being displayed, turn the Time/div knob clockwise to raise the sampling rate.
- Remove components from the observed signal that have frequency higher than the Nyquist frequency. Apply Bandwidth Limiting filters or change the Coupling (both done on the CH setup menu) to remove high frequency components, or use an external filter.

Frequency Range of FFT Waveform

With digital oscilloscopes, unlike with dedicated spectrum analyzers, the frequency range cannot be freely set.

The frequency axis range of FFT waveforms is determined based on current sampling rate, memory length, and horizontal axis setups.

The frequency axis range is from 0 Hz (DC) to the Nyquist frequency (half of the FFT sampling rate frequency).

Math Trace Scale

When performing a Math operation which has two source inputs, the resulting Math operation data is calculated using the larger vertical scale of the two source waveforms. The source with the smaller vertical scale is calculated as a ratio that is applied to source waveform.

Example: Addition of CH1 = 2V/div and CH2 = 10 mV/div
Math operation data = CH1 + CH2 × 0.01 / 2

When displaying Math results, set a range that will enable the Math waveform data to fit within the screen. Afterward, this Math waveform data range can be manually adjusted as necessary.

Table 20: Vertical Range for Math Display

		CH1 range											
	V	10V	5V	2V	1V	0.5V	0.2V	0.1V	50mV	20mV	10mV	5mV	2mV
CH2 range	10V	10	10	10	10	10	10	10	10	10	10	10	10
	5V	10	5	5	5	5	5	5	5	5	5	5	5
	2V	10	5	2	2	2	2	2	2	2	2	2	2
	1V	10	5	2	1	1	1	1	1	1	1	1	1
	0.5V	10	5	2	1	0.5	0.5	0.5	0.5	0.5	0.5	0.5	0.5
	0.2V	10	5	2	1	0.5	0.2	0.2	0.2	0.2	0.2	0.2	0.2
	0.1V	10	5	2	1	0.5	0.2	0.1	0.1	0.1	0.1	0.1	0.1
	50mV	10	5	2	1	0.5	0.2	0.1	50m	50m	50m	50m	50m
	20mV	10	5	2	1	0.5	0.2	0.1	50m	20m	20m	20m	20m
	10mV	10	5	2	1	0.5	0.2	0.1	50m	20m	10m	10m	10m
	5mV	10	5	2	1	0.5	0.2	0.1	50m	20m	10m	5m	5m
	2mV	10	5	2	1	0.5	0.2	0.1	50m	20m	10m	5mV	2m

Adjust Math Trace Range

The Math trace has its own OFFSET and Range knobs, similar to each Channel trace.

The Math **OFFSET knob** controls the Math trace Vertical offset. Turn the knob clockwise to move the trace in the positive direction or counter-clockwise to move in the negative direction. Press this knob once to move the trace so that the offset is O and press it again to restore the original offset value. Offset values are shown on the Math descriptor box below the grid.

Math **Range Control knob** controls the sensitivity for the Math trace. Turn the knob clockwise to switch to a higher sensitivity range, or counter-clockwise to switch to a lower sensitivity range. The range set for the math trace is shown on the Math descriptor box below the grid.

Relation to Other Functions

The following constraints apply to FFT, derivative and integral operations:

- **Roll Mode.** FFT, derivative and integral operations cannot be displayed at Roll Mode. If Roll Mode is turned on while the Math trace is displayed, it will appear on the Horizontal menu that Roll Mode has been set, but when you adjust the timebase to 50 ms/div or slower to start Roll Mode, the Math trace will disappear. However, if you then set a value of faster than 50 ms/div with the Time/div knob, the FFT, derivative and integral waveforms will appear again.
- **Auto Measure.** The Auto Measure function cannot be applied to FFT, derivative, and integral operation Math traces.

Reference Waveforms (Memories)

The Reference function saves the displayed waveform data and setups to internal memory locations so they can be easily recalled to the screen.

Recalled reference waveforms are displayed as a white trace. They can be shown next to other active traces.

Use the Save/Recall function to export a Reference Waveform to a USB drive for long-term storage and sharing.

Save Reference

1. Press the **REF button** to open the Reference Waveform menu.
2. Continue touching Ref. No. to select the reference location, REF 1 to REF 5.

   NOTE: The new data will overwrite whatever is currently saved in that location. There is no way to manually clear the data.
3. To display the reference waveform immediately after saving (and whenever touching REF), touch **Waveform** until On is selected. Otherwise, you may leave this option Off.

   NOTE: In some cases, when pressing REF to save a new reference, another reference waveform will appear on the screen. This is because Waveform has been set to On for that reference. If it is not needed, select the Ref. No. and turn Waveform Off.
4. Touch **Source** and select the waveform to save.
5. Touch **Save**.

Recall Reference

1. Press the **REF button** to open the Reference Waveform menu.
2. Continue touching Ref. No. to select the reference location, REF 1 to REF 5.
3. Touch **Waveform** until **On** is selected.
4. Touch **Recall REF Setup**.

Relation to Other Functions

- **Zoom.** When the Zoom function is turned on, the Zoom window's data is saved to the Reference, not the source waveform in the Main window. It is not possible to further Zoom reference waveforms.
- **XY Display.** XY traces cannot be saved to Reference. The REF function is inactive during XY display.
- **Roll Mode.** Roll Mode stops during processing of reference waveforms. It is resumed when acquisition resumes after the processing ends.

Help

The oscilloscope includes an onboard Help system explaining the use of certain controls. To get Help:

1. Press the **HELP button** in the upper left of the front panel. You will briefly see the message, "Display help message of selected item." in the Message bar.
2. **Operate the control** for which you'd like Help (press the button or turn the knob). The Help topic appears on the touch screen. Scrolling is not required.
3. Press the **Close Menu button** to exit Help.

NOTE: Pressing HELP again does not close the Help function, although it will close the topic display.

Clear

WaveJet Touch has a Clear function that can reinitialize the following acquired waveform data:

- Acquisition history (i.e., sweeps available for Replay)
- Average processing
- Persistence data (i.e., sweeps prior to last shown on display)
- Measurement results
- Measurement statistics
- Pass/Fail counts and judgments
- Log entries

CAUTION. The data listed above is deleted all at once when the front panel CLEAR button is pressed. After Clear, it cannot be restored. Execute Clear only after necessary data is recorded.

Display

The Display settings control the appearance of waveform traces, including trace type, vector (line), intensity (brightness), persistence, and color gradation.

Type

By default, channel waveforms are set to display as YT (Voltage vs. Time) traces. To change this to an XY trace, which displays the phase relationship between the inputs:

1. Attach your signals to **CH1** and **CH2**.

   NOTE: When XY display is selected, CH1 is used as the X input and CH2 is used as the Y input. CH3 and CH4 are disabled.

2. Touch the front panel **DISPLAY** button to open the Display menu.
3. Touch **Type** to open the Type sub-menu.
4. Choose **XY** or **XY (Triggered)**.

   NOTE: When XY is selected the display will update regardless of the trigger condition. When XY (Triggered) is selected the display will update only when a trigger condition is met.

Vector

By default, the oscilloscope interpolates points between actual sample points to display a smooth lined waveform trace. This line can be removed to display only the sampled points.

NOTE: This option is not available for XY trace displays.

1. Touch the front panel **DISPLAY** button to open the Display menu.
2. Touch **Vector** until the icon showing the desired style is selected.

Intensity

The intensity setting controls the brightness of the waveform traces relative to the grid. It is set as a percentage; the default is 50%.

The intensity value can be set from the Display menu:

1. Touch the front panel **DISPLAY** button to open the Display menu.
2. Touch **Intensity**, then turn the **ADJUST knob** until the desired percentage is shown. A higher value is brighter, a lower value is dimmer.

Persistence

The Persistence feature retains samples on the display for a pre-set amount of time before allowing them to gradually "decay" (fade). This results in a trace that appears more like that of traditional analog oscilloscopes.

1. Touch the front panel **DISPLAY** button to open the Display menu.
2. Touch **Persistence Time** until the desired decay time is shown. You can also use the ADJUST knob to enter the value, although only these preset increments may be selected using either method: Off, 100 ms, 200 ms, 500 ms, 1 s, 2 s, 5 s, 10 s or ∞ (infinitely).

Relation to Other Functions

- **Roll Mode.** Persistence does not work when Roll Mode is on.
- **Measure.** The Persistence display is cleared and begins anew whenever measurements are turned on.

Color

By default, waveform traces display in a single color matches to the input channel. You can apply a color gradient that displays more frequent sample points in hotter colors.

1. Touch the front panel **DISPLAY** button to open the Display menu.
2. Touch **Color Gradation** until **Spectrum** is selected.

Utilities

Print

Use the Print menu to configure the print function output: hardcopy print or print to file (on a USB drive).

Send to Printer

WaveJet Touch supports PictBridge-compliant printers.

1. Press the **UTILITIES button**, then choose **Next > Remote**. Turn **OFF** remote control.
2. Press **CLOSE MENU** to return to the Utilities menu, then touch **Next** until back at the first page. Touch **Print.**
3. Touch **Device** and select **Printer**.
4. Connect the printer to the back panel USB port.
5. When you see the message “Printer is connected,” touch **Paper Size** then turn the **ADJUST knob** until the correct paper size is selected.
6. Touch **Background** and choose **Black** (like the oscilloscope screen) or **White**.
7. Press the front panel **PRINT button** once to execute the print; press it again to cancel printing.

Print to File (USB Memory)

1. Press the front panel **UTILITIES** button, then choose **Print**.
2. Touch **Device** and choose to output to **USB Memory**.
3. Insert the drive into the front panel USB port.

   TIP: To check the available space on the drive, go to Utilities > Status & Update > USB Memory Information.
4. Touch **File format** and choose the output type, **TIFF**, **BMP**, or **PNG**.
5. Touch **Background** and choose **Black** (like the oscilloscope screen) or **White**.
6. If using PNG format (only), touch **Transparency** and choose **Off** or **On** to apply transparency to the background color.
7. Touch **File Name** and enter a default name:
   - Turn the **ADJUST knob** until the desired first character is highlighted on the popup Character List; press the ADJUST knob to select it. Repeat for the remaining three alpha characters.
   - Optionally, enter numbers for the final four numeric characters.

   NOTE: The number is automatically incremented with each new capture.
8. Press the front panel **PRINT button** at any time to capture and save a file on the USB drive.

CAUTION. When copying data to or from a USB drive, do not remove the device if it is still blinking. Doing so may cause the loss of screen and other data.

Configuration

Config. opens a large sub-menu containing many oscilloscope operational settings.

Language

To change the language of the touch screen display:

1. Press the **UTILITIES button**, then touch **Config**.
2. Touch **Language** to open the Language sub-menu.
3. Select a language.

Date & Time

To set the clock:

1. Press the **UTILITIES button**, then touch **Config.**
2. Touch **Date & Time** to open the Date & Time sub-menu.
3. Touch **Display** and choose to display the **Real-time Clock** or the **Acquisition Time-stamp** in the Message bar. **Off** disables the clock display.
4. Touch **Year** and enter the year.
5. Touch Month/Day until the **Month** is selected, then turn the **ADJUST knob** until the number of the month appears. Touch Month/Day again so the **Day** is selected, then turn the **ADJUST knob** until the date appears.
6. Touch Hour/Min until the **Hour** is selected, then turn the **ADJUST knob** until the number of the hour appears. Touch Hour/Min again so the **Min** is selected, then turn the **ADJUST knob** until the minute appears.
7. Touch **Enter**.

Offset Setting

How a waveform appears to enlarge/shrink on the grid when Volts/div is adjusted depends on the Offset Setting constant.

1. Press the **UTILITIES button**, then touch **Config.**
2. Touch **Offset Setting** until the desired choice is selected:
   - **Division** enlarges/shrinks the waveform relative to its own GND position. The channel-specific GND position does not change when Volts/div is adjusted, which makes it useful for comparing two waveforms.
   - **Volts** enlarges/shrinks the waveform relative to center of screen (point 0 on the grid). Parts of the waveform that "disappear" off the grid when the Vertical range is increased can be seen by adjusting the Vertical Offset.

Delay Setting

The Delay Setting constant is the Horizontal axis equivalent of Offset Setting. It controls how waveforms expand/contract on the grid when Time/div is adjusted.

1. Press the **UTILITIES button**, then touch **Config.**
2. Touch **Delay Setting** until the desired choice is selected:
   - **Division** enlarges/shrinks the waveform horizontally relative to trigger point. This selection fix the trigger point and will always keep waveforms centered on the trigger point.
   - **Time** enlarges/shrinks the waveform horizontally relative to center of screen (point 0 on the grid). This selection will fix the delay time and will always keep waveforms centered in time but can cause the trigger point to move off screen.

Beep

This setting enables/disables oscilloscope sound effects. When Beep is "on," you will hear a small beep every time a control is touched/pressed, and when any other event for which you have selected audible warnings (such as Pass/Fail) occurs.

1. Press the **UTILITIES button**, then touch **Config.**
2. Touch **Next** to display Page 2/3.
3. Touch **Beep** until the Off (red X) or On icon is selected.

Panel Lock

Panel Lock disables front panel and touch screen controls. It is invoked automatically when the oscilloscope is being operate remotely.

1. Press the **UTILITIES button**, then touch **Config.**
2. Touch **Next** to display Page 2/3.
3. Touch **Panel Lock** until the Unlocked or Locked icon is selected.

Graticule

The change the grid graticule (line style):

1. Touch the front panel **UTILITIES** button to open the Utilities menu.
2. Touch **Next** to display Page 2/3.
3. Touch **Graticule**, then choose from:
   - **Grid** (default)—a traditional, fully lined grid showing all 8 Vertical division and 10 Horizontal divisions.
   - **Axis**—only the frame (outside edge) and the X and Y axes are marked, not the divisions.
   - **Frame**—only the outside edge of the grid is marked. There are no lines marking the axes or divisions.

Grid Intensity

The grid intensity controls the brightness of the display grid lines relative to the waveform trace.

1. Press the **UTILITIES button**, then touch **Config.**
2. Touch **Next** to display Page 2/3.
3. Touch **Grid Intensity**, then turn the ADJUST knob until the desired percentage is shown. A lower percentage dims the lines, a higher percentage brightens them.

Trigger Counter

This setting adds (On) or removes (Off) the trigger count from the Message bar. In the trigger counter function the frequency of the trigger source signal is measured.

1. Press the **UTILITIES button**, then touch **Config.**
2. Touch **Next** twice to display Page 3/3.
3. Touch **Trigger Counter** until the desired setting is selected.

Power Management

The oscilloscope can be configured to go into a reduced power mode after a set period with no touch screen activity.

1. Press the **UTILITIES button**, then touch **Config.**
2. Touch **Next** twice to display Page 3/3.
3. Touch **Power Management**.
4. Touch **Backlight Off** and choose the time period. After this time period from last operation the oscilloscope backlight will go off.
5. Touch **Power Off** (sleep mode) and choose the time period. After this time period from last operation the oscilloscope will power off. In order to power the oscilloscope back on press the power button in to the "off" state and then press the power button back to the "on" state.
6. Touch **Backlight Intensity** and choose High, Medium, or Low.

Calibration

Auto Calibration

This routine automatically re-calibrates the front-end (Vertical axis) gain at power on and whenever there is a ± 5° C temperature change since the last auto or self calibration.

1. Press the **UTILITIES button**, then touch **Calibration.**
2. Touch **Auto Calibration** until the desired setting is selected.

Self Calibration

This option manually starts the front-end calibration routine. Calibration takes approximately 1 minute to complete.

NOTE: Do not calibrate the oscilloscope with a signal attached to any input.

1. Press the **UTILITIES button**, then touch **Calibration.**
2. Touch **Self Calibration**.

Touch Screen Calibration

This routine improves the touch screen registration. Perform it periodically to ensure good performance. It is best to use a stylus or a capped pen for this procedure.

1. Press the **UTILITIES button**, then touch **Calibration.**
2. Touch **Touch Screen Calibration**.
3. Touch as close as possible to the center of each X that appears on the touch screen.

Status & Update

View Status

This option displays oscilloscope hardware and firmware information.

1. Press the **UTILITIES button**, then touch **Status & Update.**
2. Touch **Status** to display oscilloscope information.

View USB Memory

This option displays available memory on a USB drive.

1. Insert the drive into the front panel USB port.
2. Press the **UTILITIES button**, then touch **Status & Update.**
3. Touch **USB Memory Information**.

Update Firmware

1. Go to teledynelecroy.com/softwaredownload and download the firmware installer onto a USB drive.
2. Insert the drive into the front panel USB port.
3. Press the **UTILITIES button**, then touch **Status & Update.**
4. Touch **Update**, then touch **OK**.

NOTE: Do not power down the oscilloscope or remove the USB drive at any time during the firmware update. Wait until you see the "Update Complete" message on the oscilloscope screen.

5. After the update is complete, power cycle the instrument.

Remote Control

The oscilloscope can be operated via routines sent from a remote controller. There are several interface options:

- **TCP/IP**
- **GPIB**
- **USB**

NOTE: When the rear-panel USB connector is configured for remote control, the Print Device option is automatically switched from Printer to USB Memory.

See the *WaveJet Remote Control Manual* for instructions on making remote connections and command syntax.

Auxillary Output

The back panel Aux Out connector can be configured to output a pulse upon a trigger event or a Pass/Fail test result. The output pulse has an amplitude greater than 3 V into 1 MΩ and 0.9 V into 50 Ω. When the output pulse is set to trigger event the pulse width is around 1.5 us and when the output pulse is set to a pass/fail test result the pulse width is around 8-10 ms. Use a BNC cable to connect the auxiliary device to the oscilloscope.

Save/Recall

The Save/Recall function saves panel setups and waveform data to either to the oscilloscope's internal memory to an external memory drive for later recall.

Internal Memory

You may save all current channel setups (only) to one of five internal setup locations.

NOTE: These are different than the five REF waveform locations. All setups are saved and will be copied to the channels when recalled, but there is no saved waveform to display. Internal setups revert to the Default Setup after a firmware update.

Save Setup

1. Press the **Save/Recall button**.
2. Touch **Save Setup to Int. Memory**.
3. Choose one of the five **Setup** locations.

Recall Setup

1. Press the **Save/Recall button**.
2. Touch **Recall Setup to Int. Memory**.
3. Choose one of the five **Setups**. The date/time it was saved, or else "Default Setup," appears on the menu.

Recall Default Setup

This option allows you to restore the factory default setups on each channel.

1. Press the **Save/Recall button**.
2. Touch **Recall Default Setup**.

The following setups are retained and are not affected by this operation.

- System settings for language, date and time, sound, panel lock, grid intensity, trigger counter, power management, offset constants, delay constants, print, and remote control
- Saved panel setups
- Saved reference waveforms

USB Memory

Setups and waveform data can be saved to an external USB drive in one of several file formats. Also use this option to export/import Reference Waveforms that have been previously saved to the oscilloscope's internal memory.

NOTE: Channel and Math traces must be turned on for display via the front panel to be saved to a waveform file.

File Formats

SETUP

This option saves only the current panel setups, no waveform data. Setups can be later recalled to the oscilloscope. Files are saved with the .SET extension. The default filename is STUP0000.SET.

BINARY

This option saves channel waveform data and other information about the oscilloscope setup which allows the waveforms to be recalled at a later time, such as the time base setting. All displayed waveforms are stored as binary values. Files are saved with the .WFM extension. The default filename is WBIN0000.WFM.

ASCII

This option saves waveform data, panel setups, and acquisition time values in comma-separated value format. Either channel or Math waveform data may be saved in this format. It can be easily opened on a computer and edited using third-party software, such as Microsoft® Excel. However, ASCII data cannot be recalled to the oscilloscope for display. Files are saved with the .CSV extension. The default filename is WASC0000.CSV or MASC0000.CSV.

MATHCAD

This option saves waveform data, panel setups, and time values in Mathcad format. The saved data is space delimited and can be easily edited using third-party software. However, like ASCII data, it cannot be recalled to the oscilloscope for display. Files are saved with the .PRN extension. The default filename is WMCD0000.PRN or MMCD0000.PRN.

REFERENCE

This option allows you to select one of the existing Reference Waveforms and export it to the USB memory with a new file name. The exported file can be later recalled the oscilloscope for display. Files are saved with the .REF extension. The default file name is WREF0000.REF.

File Naming

Files names are eight characters long and can be set to any combination of four alpha characters and four digits. If not set manually, the last filename entered will automatically increment by one digit whenever a new file is saved.

Save to USB Memory

1. Insert the drive into the USB port on the front of the oscilloscope.
2. Press the front panel **Save/Recall button**.
3. Touch **Save/Delete to USB memory**, then touch **Save/Delete** until Save is selected.
4. Touch **File format** and choose one of the formats.

   NOTE: To save a Math trace, touch Next and choose Math (ASCII) or Math (Binary).
5. Optionally, touch **File Name** and enter a new filename:
   - With the first position selected**, turn ADJUST** to select a character from the Character List popup. **Push ADJUST** to enter and go to the next position.
   - Repeat for the remaining positions.
6. If exporting a Reference Waveform, touch **Ref. No.** until the desired REF is selected.
7. Touch **Save**.

CAUTION. Do not remove the USB drive until it has stopped blinking and the message “XXXXYYYY (file name)” disappears from the Message bar.

Recall from USB Memory

1. Insert the drive into the USB port on the front of the oscilloscope.
2. Press the front panel **Save/Recall button**.
3. Touch **Recall from USB memory**.
4. Touch **File format** and select the format of the file to be recalled.
5. Touch **File List**, then **turn ADJUST** to select the file. Ensure that the Rotate setting is set to **Select File**.

   NOTE: Up to 20 files may be displayed at once. When there are more than 20 files, continue to turn ADJUST to scroll to the next page or press ADJUST so that Select Page is the current Rotate setting.
6. Touch **Recall**.

Delete from USB Memory

1. Insert the drive into the USB port on the front of the oscilloscope.
2. Press the front panel **Save/Recall button**.
3. Touch **Save/Delete to USB memory**, then touch **Save/Delete** until Delete is selected.
4. Touch **File format** and choose the format of the file to be deleted.
5. Touch **File List**, then **turn ADJUST** to select the file. Ensure that the Rotate setting is set to **Select File**.

   NOTE: Up to 20 files may be displayed at once. When there are more than 20 files, continue to turn ADJUST to scroll to the next page, or press ADJUST so that Select Page is the current Rotate setting.
6. Touch **Delete File**.

Reformat USB Memory

This option deletes all files as it reformats the drive.

1. Insert the drive into the USB port on the front of the oscilloscope.
2. Press the front panel **Save/Recall button**.
3. Touch **Save/Delete to USB memory**, then touch **Save/Delete** until Delete is selected.
4. Touch **Format USB Memory**.

Maintenance

This chapter describes daily maintenance and calibration methods that help ensure precise measurements.

Cleaning

Before cleaning this oscilloscope, unplug its power cord to prevent possible electric shocks. Lightly wipe the exterior with a soft cloth moistened with water or diluted neutral detergent. Use of a solution or detergent that is inappropriate for cleaning may cause product discoloration or an unexpected problem may occur.

• Solution or detergent that may be used: Water, neutral detergent (diluted)

• Solution or detergent that must not be used: Alcohol, gasoline, acetone, lacquer, ether, thinner, ketone-based detergent

Calibration

This oscilloscope is equipped with Self and Auto Calibration routines. See procedures on p. 70.

These calibration routines are capable of maintaining the usual precision of the oscilloscope for a certain amount of time, only. The guideline is to perform a factory calibration regularly after approximately 2000 hours of continuous use, which usually works out to about one calibration per year.

Auto Calibration

The Auto Calibration routine calibrates the AD converter full-scale, offset and straight line. An initial calibration is always performed when the oscilloscope is first powered on. If Auto Calibration is set to On, calibration is repeated whenever there is a temperature change of 5° C.

NOTE: Persistence display is cleared whenever Auto Calibration is performed. To prevent this, turn OFF Auto Calibration.

Self Calibration

Self Calibration calibrates the vertical axis range, horizontal axis range, offset, and trigger level. About a minute is needed to complete this calibration. When finished, the message "Self Calibration completed." is displayed in the Message bar.

If Self Calibration does not end normally, the message "Self Calibration failed." is displayed. Contact Technical Support after several failed attempts.

Requests for Repair or Calibration

Teledyne LeCroy warrants this oscilloscope for normal use and operation within specification for a period of one year from the date of shipment. Spare parts, replacement parts and repairs are warranted for 90 days.

In exercising its warranty, Teledyne LeCroy, at its option, will either repair or replace any assembly returned within its warranty period to the Customer Service Department or an authorized service center. However, this will be done only if the product is determined by Teledyne LeCroy's examination to be defective due to workmanship or materials, and the defect is not caused by misuse, neglect, accident, abnormal conditions of operation, or damage resulting from attempted repair or modifications by a non-authorized service facility.

The customer will be 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 transportation charges prepaid.

Returning an Oscilloscope

Contact your local Teledyne LeCroy sales representative to find out where to return the product. All returned products should be identified by model number and serial number. Provide your name and contact number and if possible describe the defect or failure. In case of products returned to the factory, a Return Authorization Number (RAN) must be used. Contact your nearest Teledyne LeCroy office to receive a RAN.

Return shipment should be prepaid. Teledyne LeCroy cannot accept COD or Collect Return shipments. We recommend air-freighting.

1. Contact your local Teledyne LeCroy sales or service representative to obtain a Return Authorization Number.
2. Remove all accessories from the probe. Do not include the manual.
3. Pack the probe in its case, surrounded by the original packing material (or equivalent) and box.
4. Label the case with a tag containing:
    - The RAN
    - Name and address of the owner
    - Probe model and serial number
    - Description of failure
5. Package the probe case in a cardboard shipping box with adequate padding to avoid damage in transit.
6. Mark the outside of the box with the shipping address given to you by the Teledyne LeCroy representative; be sure to add the following:
    - ATTN: <RAN assigned by the Teledyne LeCroy representative>
    - FRAGILE
7. Insure the item for the replacement cost of the probe.
8. **If returning a probe to a different country, also**:
    - Mark shipment returned for service as a "Return of US manufactured goods for warranty repair/recalibration."
    - If there is a cost involved in the service, put the service cost in the value column and the replacement value in the body of the invoice marked "For insurance purposes only."
    - Be very specific as to the reason for shipment. Duties may have to be paid on the value of the service.

Storage and Transportation

Do not store the product where it will be subject to any of the following.

- Direct sunlight
- Dust
- Corrosive gas
- Temperatures outside the operating range of −20 to +60°C
- Humidity outside the operating range of 5% to 80% RH (no condensation)

When transporting the product, use the carton box in which the product was delivered or a box with equivalent packing materials.

Troubleshooting Guide

The procedures described in Table 21 can be safely performed by end users. Please consult the table before contacting Technical Support.

If the oscilloscope still does not operate normally after troubleshooting, please contact your nearest Teledyne LeCroy service center at the address listed in this manual.

Table 21: Troubleshooting Guide

Problem	Check	Action
Does not start when POWER is switched on.	Is the power cord plugged into an AC outlet?	Plug it into an AC outlet.
	Is the power cord connected to the AC power input terminal on the rear panel?	Connect it to the AC power input terminal.
	Is the fan motor running (on the left side this product)?	Place your hand next to the ventilation holes. If you do not feel any air movement, the fan is not running. If the fan is running but the product will not start, it may be due to a broken backlight. Contact Technical Support.
After switching on POWER, the power suddenly shuts off.	This oscilloscope is equipped with an overheating protection function.	
	Is the ambient temperature in the operating temperature range of 0°C to 40°C?	Only use this product when the ambient temperature is within the operating temperature range. Set the power switch to Standby for a while before setting it ON.
	Is anything obstructing the holes on the left and right sides that are used by the ventilation fan?	After removing obstructions, set the power switch to Standby for a while before setting it ON.
Screen is hard to see due to poor contrast.	Is the ambient temperature in the operating temperature range of 0°C to 40°C?	Only use this product when the ambient temperature is within the operating temperature range.
	Has waveform or grid intensity been set too low?	Press the DISPLAY button, touch Waveform Intensity, then turn the ADJUST knob to increase trace brightness. Press the UTILITIES button, touch Config → Grid Intensity, then turn the ADJUST knob to increase grid brightness.
	Has backlight intensity been set too low?	Press the UTILITIES button, touch Config → Power Management → Backlight Intensity and select Medium High.
After signal input, there is no waveform display.	Are the Vertical, Horizontal, and Trigger setups correct?	Press the AUTO SETUP button.
	Is the channel turned on?	Connect an input signal, then press the CH menu button.
	Is the probe damaged?	If the probe is damaged, replace it.

Problem	Check	Action
Synchronization does not work.	Was the sync signal selected correctly?	Press the TRIGGER Setup button, then select the sync signal source under the Source menu item.
	Is the trigger level within the waveform amplitude range?	Make sure the trigger level mark on the right side of the screen (T) is within the waveform amplitude range. Press or turn the Trigger LEVEL control knob until the trigger level mark is inside the range.
The auto setup for probe ratio does not work correctly.	Does the probe used have a probe sensor function?	Use a probe that has a probe sensor function.
Product reboots repeatedly and cannot measure.	Is the AC supply voltage too low?	Make sure the AC supply voltage is within the rated range.
The clock display in the lower right part of the screen is inaccurate.	Was the clock set correctly?	Press UTILITIES button, touch Config. → Date & Time, and set the clock correctly. If the clock is still not accurate, it may be due to a depleted internal battery. Contact Technical Support.
Panel setups prior to last power-off are not restored when power is turned back on.		Internal battery may be depleted. Contact Technical Support.
Auto Calibration does not work correctly. Self Calibration does not work correctly.		Press the UTILITIES button and touch Calibration. Set Auto Calibration to ON. Disconnect all input signals, then run Self Calibration. If calibration still does not work after trying these two actions, contact Technical Support.

Contact Us

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: 800-909-7211 / 408-727-6600
contact.corp@teledynelecroy.com
Support: 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: +44 12 765 03971

Singapore

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

China

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

700 Chestnut Ridge Road
Chestnut Ridge, NY 10977
USA
www.teledynelecroy.com