Welcome to the 1 Series DPO Help

You can scan the QR code below to access the latest version of this Help/User manual online at tek.com/docs/1series.

This Help supports the following products.

Model	Description
DPO1052	2 Channel, 50 MHz
DPO1072	2 Channel, 70 MHz
DPO1102	2 Channel, 100 MHz
DPO1202	2 Channel, 200 MHz
DPO1054	4 Channel, 50 MHz
DPO1074	4 Channel, 70 MHz
DPO1104	4 Channel, 100 MHz
DPO1204	4 Channel, 200 MHz
DPO1052S	2 Channel, 50 MHz with AFG
DPO1072S	2 Channel, 70 MHz with AFG
DPO1102S	2 Channel, 100 MHz with AFG
DPO1202S	2 Channel, 200 MHz with AFG
DPO1054S	4 Channel, 50 MHz with AFG
DPO1074S	4 Channel, 70 MHz with AFG
DPO1104S	4 Channel, 100 MHz with AFG
DPO1204S	4 Channel, 200 MHz with AFG

Open Source GPL License Notice

For programs licensed under the "GNU General Public License (GPL) or Lesser GNU General Public License (LGPL)" the complete corresponding sources are available. You can order a CD containing the sources from us for a period of three years after download of the software, by sending a written request to:

Chief Intellectual Property Counsel, Tektronix, Inc.

MS 50/LAW

13725 SW Karl Braun Dr.

Beaverton OR, 97077

This offer is valid to anyone in receipt of this information.

Your request should include: (i) the name of the product, (ii) your (company) name, and (iii) your return mailing and email address (if available).

Please note that we may charge you a fee to cover the cost of performing this distribution.

All the copyright and license information of open source packages used in the product software are available in attribution.zip present in C drive of instrument.

Instrument compliance markings

The following table list the compliance markings that apply to the 1 Series DPO instruments.

Compliance labels		Description
CE	CE MARK	CE marking is a certification mark that indicates conformity with health, safety, and environmental protection standards for products sold within the European Economic Area.
	WEEE MARK	WEEE marking indicating separate collection for WEEE- Waste of electrical and electronic equipment. The mark is required per Article 11(2) of the WEEE Directive.
TB	TB RoHS MARK	TB RoHS marking indicates that the item is certified by Tektronix to be RoHS Compliant to the 10 restricted substances according to EU Directive 2015/863 and that Tektronix has published a TDoC and/or DoC.
40	CHINA RoHS 40 MARK	China RoHS 40 marking is a certification mark that indicates conformity with the Chinese government regulation to control the restriction of hazardous substances for products shipped to China.
	RCM MARK	RCM (Regulatory Compliance Mark) marking is a certification mark that signifies product compliance with electrical safety, EMC, EME & telecommunications for Australia and New Zealand.
UK CA	UKCA MARK	UKCA (UK Conformity Assessed) marking is a new UK certification mark that indicates conformity with health, safety, and environmental protection standards for products sold within Great Britain (England, Wales and Scotland). It covers most goods which previously required the CE marking.

	UKRAINE MARK	Ukraine marking is the certification mark that indicates conformity to applicable technical regulations for products sold within the Ukraine.
Mfr. Compliance Contact Tektronix, Inc. PO Box 500, MS 19-045 Beaverton, OR 97077, USA www.tektronix.com	COMPLIANCE CONTACT ADDRESS MARK TEK	Compliance Contact Address marking is the point of contact address for Tektronix Compliance department.
KCC-REM-tek-OSC	KCC-REM-tek-OSC MARK	KC (Korea Certification) mark signifies compliance with Korea's product safety requirements for electrical and electronic equipment / Tektronix Major Product Type: Oscilloscopes / Tektronix Product ID Code: KCC-REM-tek-OSC.

Product documents and support

Finding documentation for your product and contacting Tektronix for support and feedback.

Product support and feedback

Tektronix values your feedback on our products. To help us serve you better, please send us your suggestions, ideas, or comments on your instrument, application, or product documentation.

Contact through mail, telephone, or tek.com.

When you contact Tektronix Technical Support, please include the following information (be as specific as possible):

General information

- All instrument model numbers
- Hardware options, if any
- Probes used
- Your name, company, mailing address, phone number, and FAX number
- Please indicate if you would like to be contacted by Tektronix about your suggestion or comments.

Application specific information

- Software version number
- Description of the problem such that technical support can duplicate the problem
- If possible, save and send the setup files for all the instruments used and the application
- If possible, save and send status messages text files
- If possible, save and send the waveform on which you are performing the measurement as a .wfm file

Product Documentation

Review the following user documents before installing and using your instrument. These documents provide important operating information.

Product documentation

The following table lists the primary product specific documentation available for your product. These and other user documents are available for download from tek.com. Other information, such as demonstration guides, technical briefs, and application notes, can also be found at tek.com. **Document**	**Content**
Help (primary user manual, this document)	In-depth operating information for the product. Available from the Help icon in the product UI and as a PDF on tek.com.
Safety and Compliance Information	Safety, compliance, basic power and environmental specifications, and instrument connector information.
Specifications and Performance Verification Technical Reference	Instrument specifications and performance verification instructions for testing instrument performance.
Programmer Manual	SCPI commands for remotely controlling the instrument.
Declassification and Security Instructions	Information about the location of memory in the instrument. Instructions for declassifying and sanitizing the instrument.

How to find your product documentation

1. Go to tek.com.
2. Click **Download** in the green sidebar on the right side of the screen.
3. Select **Manuals** as the Download Type, enter your product model, and click **Search**.
4. View and download your product manuals. You can also click the Product Support Center and Learning Center links on the page for more documentation.

Copyright © 2026, Tektronix, Inc. All rights reserved. Licensed software products are owned by Tektronix or its subsidiaries or suppliers, and are protected by national copyright laws and international treaty provisions. Tektronix products are covered by U.S. and foreign patents, issued and pending. Information in this publication supersedes that in all previously published material. Specifications and price change privileges reserved.

TEKTRONIX and TEK are registered trademarks of Tektronix, Inc.

These are the original instructions in English.

Go to tek.com/eula to read the Tektronix End User License Agreement.



Contacting Tektronix

Tektronix, Inc.
13725 SW Karl Braun Drive
P.O. Box 500
Beaverton, OR 97077
USA

For product information, sales, service, and technical support:

In North America, call 1-800-833-9200.

Worldwide, visit www.tek.com to find contacts in your area.

Contents

Welcome to the 1 Series DPO Help **8**
- Open Source GPL License Notice 8
- Instrument compliance markings 9
- Product documents and support 10
  - Product support and feedback 10
  - Product Documentation 10

Installing your instrument **12**
- Operating requirements 12
- Input signal requirements 13
- Check that the instrument passes power-on self tests 13
- Connecting probes to the instrument 13
- Rackmount option information 13

Getting acquainted with your instrument **14**
- Front panel controls and connectors 14
- Button and knob functions 15
- Rear panel connections 17
- User interface 18
- User interface elements 19
- Badges 21
  - Channel and Waveform badges 21
  - Measurement badges 22
  - Wide badge 22
  - Mask Test badge 23
  - Cursor badges 24
  - Deleting mass Measurements badges at once 24
  - Signal clipping and badges 25
  - System badges 25
  - Common badge actions 25
  - Badge selection status 26
- Configuration menus 26
- Zoom user interface 27
- Using the touch screen interface for common tasks 28
- Accessing application help 29

Configure the instrument **30**
- Download and install the latest instrument firmware 30
- Set the time zone and clock readout format 30
- Functional check 31
- Run Signal Path Compensation (SPC) 31

Compensate the probe . . . 32
Connect to a network (LAN) . . . 33
Connect the oscilloscope to a PC using a USB cable . . . 33
Connect a mouse . . . 33
ESD Prevention Guidelines . . . 33

Analog channel operating basics . . . 35
Acquiring a signal . . . 35
Autoset to quickly display a waveform . . . 36
Set Horizontal parameters . . . 37
How to trigger on a signal . . . 37
Set the acquisition mode . . . 38
Start and stop an acquisition . . . 39
Add a channel waveform to the display . . . 40
Configure channel or waveform settings . . . 40
Add a math, reference, or bus waveform . . . 41
Add a measurement . . . 42
Configure a measurement . . . 45
Delete a Measurement badge . . . 47
Display an XY plot . . . 48
Display an FFT math waveform . . . 48
Add a serial bus to the Waveform view . . . 49
Change waveform view settings . . . 50
Display and configure cursors . . . 50
Using Default Setup . . . 52

Advanced triggering . . . 53
Trigger on a pulse width event . . . 53
Trigger on a timeout event . . . 53
Trigger on a runt signal event . . . 54
Trigger on a rise or fall time event . . . 55
Trigger on a serial bus . . . 56

Setting waveform display parameters . . . 57
Set the Waveform Interpolation mode . . . 57
Set the waveform persistence, style, and intensity . . . 57
Set the graticule style and intensity . . . 57

Mask testing . . . 59
Create a Mask on waveforms . . . 59

Zoom in on waveforms . . . 60
Turn on Zoom mode . . . 60

Measurement configuration . . . 60
Label a measurement . . . 60
Set measurement reference levels . . . 61
Set measurement gates . . . 62

Saving and recalling information **63**
Save a screen image 63
Save a waveform or spectrum trace to a file 63
Save instrument settings to a file 64
Recall a reference waveform 65
Recall a Setup file 65

Menus and dialog boxes **67**
Add Measurements configuration menu overview 67
Standard measurements tab 67
Amplitude Measurements panel 68
Timing Time Measurements panel 69
The PowerFRA tab 71
Control Loop Response (Bode) measurement configuration panel 71
Configure Profile for Frequency Response Analysis (FRA) measurements 73
Measurement configuration menu overview 74
Measurement Name panel (Measurement configuration menu) 75
Configure panel (Measurement configuration menu) 75
Reference Levels panel (Measurement configuration menu) 76
Gating panel (Measurement configuration menu) 77
Add Plot configuration menu 78
Add Results Table configuration menu 78
Results Tables operations overview 78
Harmonics/Frequency Response Analysis (FRA) Results table configuration menu 79
Save As configuration menu, Custom Results Table 79
Measurement Results Table configuration menu 80
Save As configuration menu (Measurement Results Table) 81
Bus Decode Results table configuration menu 82
Save As configuration menu (Bus Decode Results table) 82
Bus configuration menu 83
I2C serial bus configuration menu 84
SPI serial bus configuration menu 85
RS232 serial bus menu 86
Analog Channel configuration menu 87
Invert channel guidelines 88
AFG configuration menu 89
Cursor configuration menu 91
Date and Time configuration menu 93
DVM configuration menu 94
Horizontal configuration menu 95
DRAW A BOX Menu 96
Mask Definition Segment configuration menu 96
Mask Test badge configuration menu 97
Right click menu functions associated with the Mask Test badge 98
Math configuration menu overview 98

Math configuration menu 99
Equation Editor (Math configuration menu) 100
Add Functions (math Equation Editor) 101
Plot configuration menus 102
Plot XY configuration menu 102
Save As configuration menu (plot Save panel, Save Plot Image button) 103
Save As configuration menu (plot Save panel, Save Plot Data button) 104
Control Loop Response (Bode) plot configuration menu 105
Reference waveform configuration menu 106
Recall configuration menu (Ref waveform configuration menu) 107
Trigger configuration menu overview 108
Edge Trigger configuration menu 108
Pulse Width Trigger configuration menu 109
Timeout Trigger configuration menu 110
Runt Trigger configuration menu 111
Rise/Fall Time Trigger configuration menu 112
Bus Trigger configuration 112
I2C serial bus trigger settings panel 113
RS-232 serial bus trigger settings panel 114
SPI serial bus trigger settings panel 114
Virtual Keyboard 115
Bus trigger radix-specific virtual keypads 115
Virtual Keypad 116
Waveform View configuration menu 116

Waveform acquisition 118
Acquisition concepts 118
Acquisition modes 119
Waveform sample interpolation 119
Coupling 120
Scaling and positioning 120
Vertical acquisition considerations 120
Horizontal acquisition considerations 121

Triggering concepts 122
Trigger sources 123
Trigger types 123
Trigger modes 123
Trigger coupling 123
Trigger slope and level 124
Trigger position in waveform record 124
Bus triggering concepts 124
Pulse width trigger concepts 124

Timeout trigger concepts . . . 124
Runt trigger concepts . . . 125
Rise/Fall time trigger concepts . . . 125

Waveform display concepts . . . 126
Waveform preview mode . . . 126
Horizontal position and the horizontal reference point . . . 126

Installing your instrument

Operating requirements and setup information to begin using your instrument.

Options

Option upgrades are not available for this product. Options need to be selected at time of purchase.

Operating requirements

Use the instrument within the required operating temperature, power, altitude, and signal input voltage ranges to provide the most accurate measurements and safe instrument operation.

Environment requirements

Characteristic	Description
Operating temperature	0°C to +50°C (+32°F to 120°F)
	For proper cooling, keep the rear of the instrument clear of obstructions for 2 inches (51 mm).
Operating humidity	> +30 °C to +50 °C: 5% to 60% relative humidity ≤ +30 °C: 5% to 90% relative humidity
	5% to 60% relative humidity at temperatures greater than +30°C and up to +50°C.
Operating altitude	Up to 3000 meters (9842 feet).

Power requirements

Characteristic	Description
Power source voltage	100 V - 240 V
Power source frequency	50/60 Hz
Power consumption	45 W maximum

Input requirements

Input	Description
Keep the input signals within allowed limits to ensure the most accurate measurements and prevent damage to the analog probes or instrument.	

Analog input channels	135 V_{RMS} Measurement Category II

Input signal requirements

Keep the input signals within allowed limits to ensure the most accurate measurements and prevent damage to the analog probes or instrument.

Make sure that input signals connected to the instrument are within the following requirements.

Input	Description
Analog input channels, 1 MΩ setting, maximum input voltage at BNC	135 V_{RMS}
	Measurement Category II

Check that the instrument passes power-on self tests

Power-on self tests verify that all instrument modules are working correctly after power up.

1. Power on the instrument and wait until the instrument screen appears.
2. Select **Utility > Self Test** from the top-edge Menu bar to open the **Self Test** configuration menu.
3. Check that the status of all power-on self tests are **Passed**.

If one or more power-on self tests shows **Failed**:

1. Power cycle the instrument.
2. Select **Utility > Self Test**. If one or more power-on self tests still shows **Failed**, contact Tektronix Customer Support.

Connecting probes to the instrument

Probes connect the instrument to your device under test (DUT). Use a probe that best matches your signal measurement needs.

Connect a BNC passive probe or cable by pushing it onto a channel BNC bayonet connector and turn the lock mechanism clockwise until it locks.

Rackmount option information

An optional rackmount kit lets you install the oscilloscope in standard equipment racks.

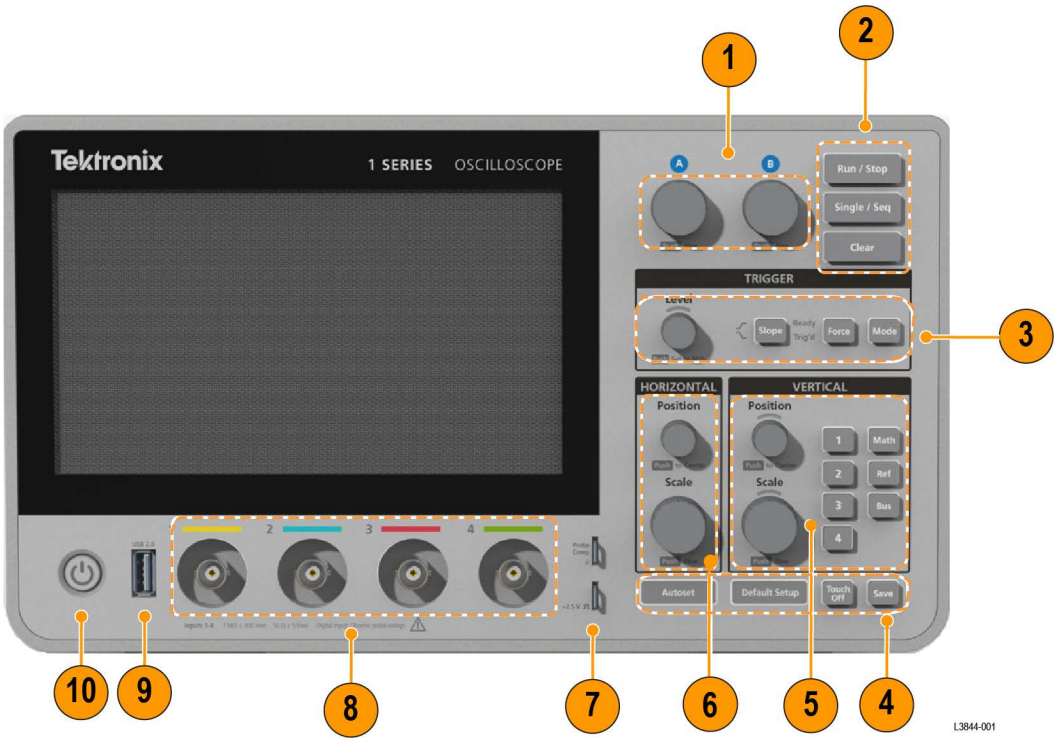
Please refer to the product datasheet at tek.com for information on rackmount options.

Getting acquainted with your instrument

The following content provides a description of the instrument controls and user interface to display waveforms and take measurements.

Front panel controls and connectors

The front panel controls provide direct access to key instrument settings such as vertical, horizontal, trigger, cursors, and zoom. The connectors are where you input signals with probes or cables.



Description	
1	Use the Multipurpose Knobs (A, B) to move cursors, adjust the zoom, and set parameter values in configuration menu input fields.
2	Use the **Acquisition** controls to start and stop waveform acquisition, enable making a single waveform acquisition, and delete the current acquisitions and measurement values from memory.
3	Use the **Trigger** controls to force a trigger event at a random point in the waveform and capture the acquisition, set the amplitude level that the signal must pass through to be considered a valid transition, and set how the instrument behaves in the absence or presence of a trigger event.
4	Use the Miscellaneous controls to turn touchscreen capability off, restore the oscilloscope settings to the default settings, automatically display a stable waveform, and save files or settings (using the current File > Save As settings).
5	Use the **Vertical** controls to move the selected waveform up or down on the screen, set the amplitude units per vertical graticule division of the selected waveform, turn on (display) or select channels, and add or select a math, reference (saved), and bus waveform on the Waveform view.

Description	
6	Use the **Horizontal** controls to move the waveform from side to side on the screen, and set the time per major horizontal graticule division and samples/second parameters for the oscilloscope.
7	Use Ground and probe compensation connectors to provide a ground connector to help reduce electrostatic damage (ESD) and adjust the high-frequency response of a passive probe.
8	Analog input channels (2 or 4, model dependant). Use the Probe connectors to connect BNC passive probes and BNC cables.
9	**USB 2.0** host port. You can connect a mouse or USB flash drive.
10	After connecting the supplied power cord, use the power button to power the instrument on and off. The power button color indicates the following instrument states; amber is standby, blue is on, unlit is off. Always allow 10 seconds between power down and power up of the instrument before operating the instrument.

Button and knob functions

A description of the functionality of each button and knob on your instrument.

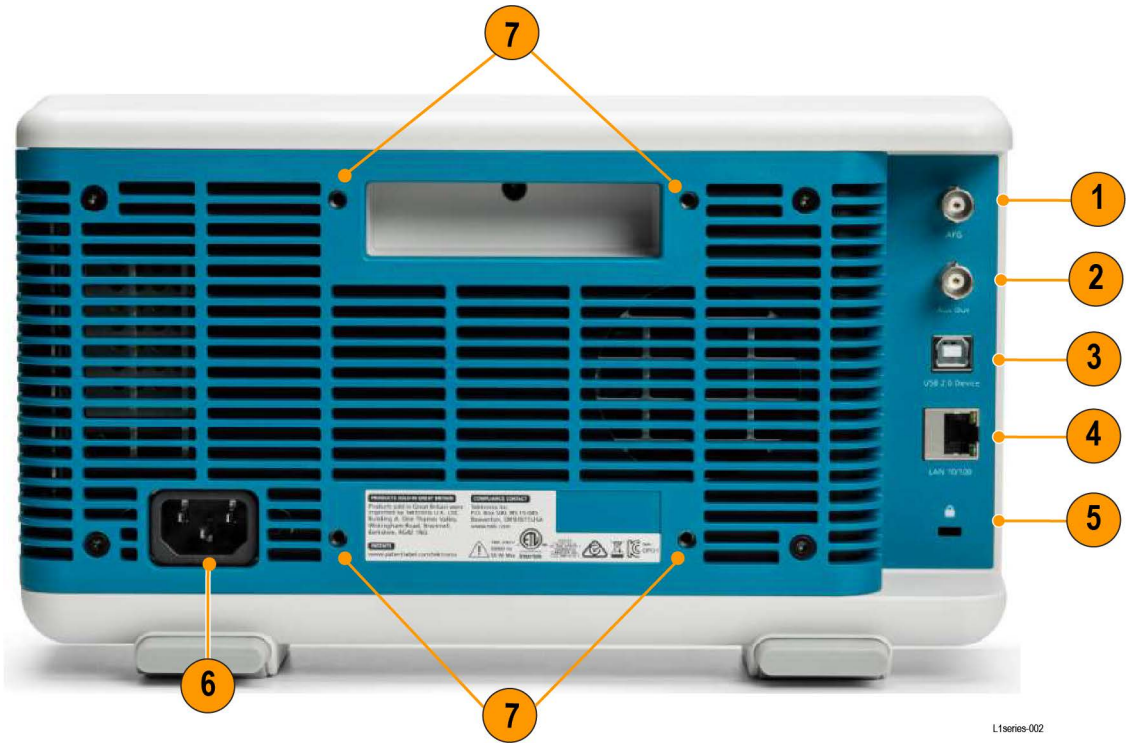
Button	Description
Run/Stop	Starts and stops waveform acquisition. The button color indicates the acquisition status (green indicates running and acquiring; red indicates stopped). When stopped, the oscilloscope shows waveforms from the last completed acquisition. The **Run/Stop** button on the screen also shows the acquisition status.
Single/Seq	Enables making a single waveform acquisition, or a specified number of acquisitions (as set in the Acquisition configuration menu). Pushing **Single/Seq** turns off **Run/Stop** mode and takes a single acquisition. The button color indicates the acquisition status (quick green flash indicates single acquisition acquired; solid green indicates waiting for trigger event). Pushing **Single/Seq** again takes another single acquisition.
Clear	Deletes the current acquisitions and measurement values from memory.
A and **B** knobs	The multipurpose knobs A and B move cursors and set parameter values in configuration menu input fields. Selecting an menu field that can use a multipurpose knob assigns the indicated knob to change the value in that input field. The arc around each knob lights when you can use that knob to do an action. Push a multipurpose knob to enable the **Fine** mode for making smaller increment changes. Push the knob again to close **Fine** mode.
Force	Forces a trigger event at a random point in the waveform and captures the acquisition.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition. The color of the **Level** knob LED indicates the trigger source except for dual-level triggers. The **Level** knob is not available when the trigger type requires two level settings or other trigger qualifiers (set from the Trigger configuration menu). Push the knob to set the threshold level to 50% of the peak-to-peak amplitude range of the signal.
Slope	Sets the signal transition direction to detect. (rising, falling, or either direction).

Button	Description
Mode	Sets how the instrument behaves in the absence or presence of a trigger event.
	Auto trigger mode enables the instrument to acquire and display a waveform whether or not a trigger event occurs. If a trigger event occurs, the instrument displays a stable waveform. If a trigger event does not occur, the instrument forces a trigger event and acquisition and displays an unstable waveform.
	Normal trigger mode sets the instrument to acquire and display a waveform only when there is a valid trigger event. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Horizontal Position	Moves the waveform and graticule side to side on the screen (changing the trigger point position in the waveform record). Push the knob to center the trigger event to the center graticule on the Waveform view.
Horizontal Scale	Sets the time per major horizontal graticule division and samples/second parameters for the oscilloscope. Scale applies to all waveforms. Push the knob to enable the **Fine** mode for making smaller increment changes. Push the knob again to close **Fine** mode.
Vertical Position	Moves the selected waveform (Channel, Math, Reference, Bus) and its graticule up or down on the screen. The color of the knob indicates which waveform the knob is controlling. Push the knob to set the threshold level to 50% of the peak-to-peak amplitude range of the signal.
Vertical Scale	Sets the amplitude units per vertical graticule division of the selected waveform. The scale values are shown on the right edge of the horizontal graticule lines. The color of the knob indicates which waveform the knob is controlling.
Channel buttons (**1,2,3,4**)	Turn on (display), select, or turn off Channel, Math, Reference, or Bus waveforms. The number of channel buttons depends on the instrument model. If the channel is not displayed, pushing a Channel button turns on that channel to the Waveform view. If the channel is on the screen and is not selected, pushing that channel's button selects that channel. If the channel is on the screen and is also selected, pushing that channel's button turns that channel off (removes it from Waveform view).
Math	Adds or selects a Math waveform on the Waveform view. If no Math waveform exists, pushing the **Math** button adds a Math waveform to the Waveform view and opens the Math configuration menu. If only one Math waveform is displayed, pushing the button turns off the Math waveform (removes it from Waveform view). Push the button again to display the waveform. If two or more Math waveforms are displayed, pushing the button cycles through selecting each math waveform.
Ref	Adds or selects a Reference (saved) waveform on the Waveform view. If no Reference waveform exists, pushing the button opens the **Browse Waveform Files** configuration menu. Navigate to and select a waveform file (*.wfm) and tap **Recall** to load and display the reference waveform. Pushing the button turns off the Reference waveform (removes it from the Waveform View). Push the button again to display the waveform.

Button	Description
Bus	Adds or selects a bus waveform on the Waveform view. If no Bus waveform exists, pushing the button adds a Bus waveform to the Waveform view and opens the Bus configuration menu. If only one Bus waveform is displayed, pushing the button turns off the Bus waveform (removes it from Waveform view). If two or more Bus waveforms are displayed, pushing the button cycles through selecting each Bus waveform.
Autoset	Automatically displays a stable waveform.
Default Setup	Restores the oscilloscope settings (such as; horizontal, vertical, scale, position) to the factory default settings.
Touch Off	Turns touch screen capability off. The button is lighted when the touch screen is turned off.
Save	Save is a one-push save operation that uses the current **File > Save As** settings to save screen-shots (including open menus and dialog boxes), waveform files, instrument settings. If a **File > Save** or **File > Save As** operation has occurred since the last instrument startup, pushing the button saves the file types to the location last set in the **Save As** configuration menu. If no file save operation has occurred since the last instrument startup, pushing the button opens the **Save As** configuration menu. Select a tab to choose the type of file to save (such as; Screen Capture and Waveform), set any associated parameters, and where to save it, and select OK. The specified file or files are saved. The next time you push the button, the same type files are saved. Screen Captures save the entire screen, including most displayed configuration menus and dialog boxes.

Rear panel connections

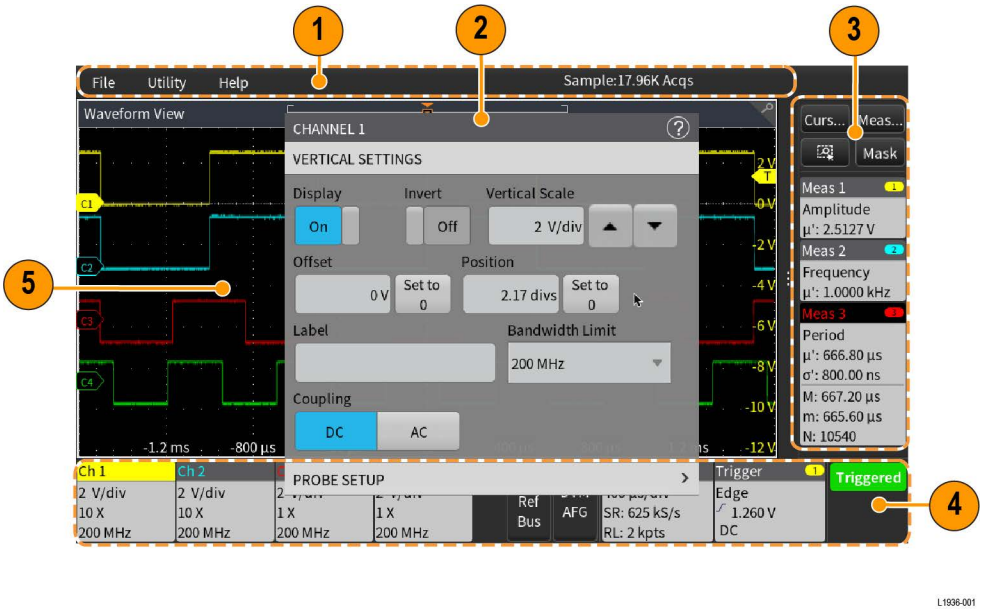
The rear panel connections supply power to the instrument and provide connectors for network, devices, and instrument mounting.



Description	
1	The **AFG** BNC connector is the signal output for the optional Arbitrary Function Generator (AFG) feature. (Select models only.)
2	The **AUX Out** BNC connector generates a signal transition on a trigger event or when output is turned on.
3	Use the **USB device** port to connect to a PC to remotely control the oscilloscope using the USBTMC protocol.
4	Use the LAN connector (RJ-45) to connect the oscilloscope to a 10/100 Base-T local area network.
5	Use the security lock connector to secure the oscilloscope to a work bench or other location with a standard PC/laptop lock cable.
6	To power the instrument, connect the supplied power cord to the power connector slot. Then connect the power cord to the appropriate AC mains source and use the power button. Use only the power cord specified for this product and certified for the country of use. To completely remove power from the instrument, disconnect the power cord.
7	Standard VESA mount compatible with off-the-shelf accessories. You can use the VESA mount (100mm x 100mm) screws to mount your instrument.

User interface

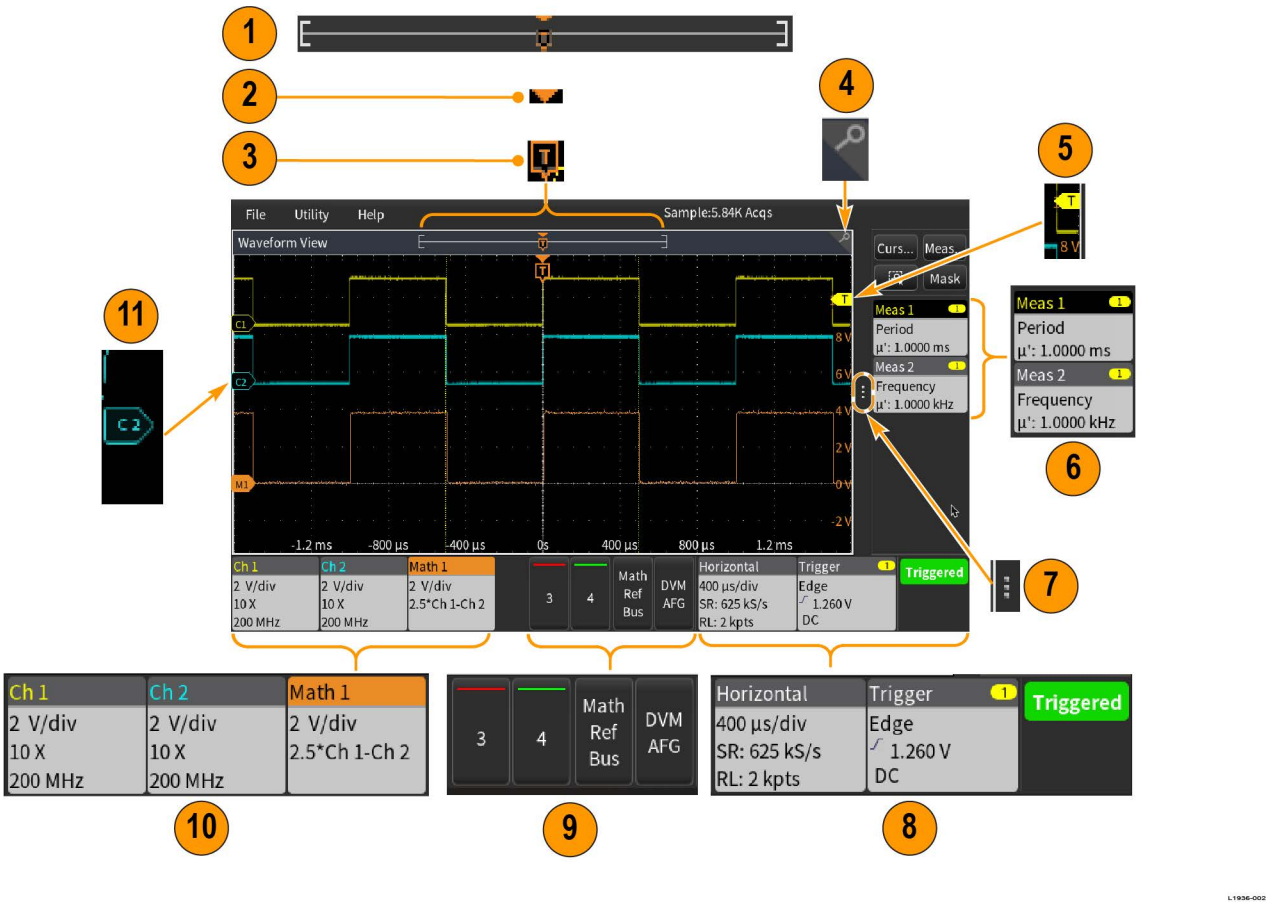
The touch screen user interface contains waveforms and plots, measurement readouts, and touch-based controls to access all oscilloscope functions.



	Description
1	The menu bar provides menus for typical operations including:
	Save, load, and access files Set oscilloscope display and measurement preferences Configure network access Run self tests Erase measurement and settings memory Open Help file
2	Configuration menus let you quickly change the parameters of the selected user interface item. You can open configuration menus by double-tapping on badges, screen objects, or screen areas.
3	The results bar contains controls for displaying measurements and cursors, adding plots, and adding result tables to the screen. This is also the Analysis Controls area where you can access controls to add items to the results bar. To remove a measurement or other badge from the results bar, flick-it off screen. The controls are:
	The **Cursors** button displays on-screen cursors in the selected view. Touch and drag, or use the multipurpose knobs, to move the cursors. Double-tap on a cursor, or on the cursor readouts, to open a configuration menu to set cursor types and related functions. The **Measure** button opens a configuration menu from which to select and add measurements to the results bar. Each measurement you add has a separate badge. Double-tap a measurement badge to open its configuration menu. The **Zoom** icon button lets you draw a box around an area of interest on the screen to zoom into. The **Mask** button allows you to access the Mask features.
4	The settings bar contains the following elements. Tap a channel or waveform button to add it to the screen and display a badge. Double-tap a badge to open its configuration menu.
	System badges for setting Horizontal, Trigger, and Date/Time parameters Inactive Channel buttons to turn on channels Add New Waveform buttons to add math, reference, and bus waveforms to the display Channel and Waveform badges that let you configure the individual waveform parameters
5	The Waveform view area displays analog, math, reference, bus, and waveforms. The waveforms include waveform handles (identifiers), individual vertical graticule scale labels, and trigger position and level indicators. You can add Measurement Results views (plots) for individual measurements. These plot views are separate view windows that you can move on the screen by dragging their title bar to a new position.

User interface elements

Each area of the user interface has a specific function that helps manage information or controls.



1. The Waveform Record View is a graphical high-level view of the overall waveform record length, how much of the record is on the screen (shown in brackets), the location of key time events including the trigger event, and the current position of waveforms cursors.

   If you are displaying a Reference waveform that is shorter than the current acquisition record length, or you are changing the horizontal time scale while the oscilloscope acquisition is stopped, the brackets change position to show the part of the waveform record that is being viewed relative to the current acquisition total record length.

   If cursors are active on a waveform, the Waveform Record View shows the relative cursor positions as small vertical dashed lines.

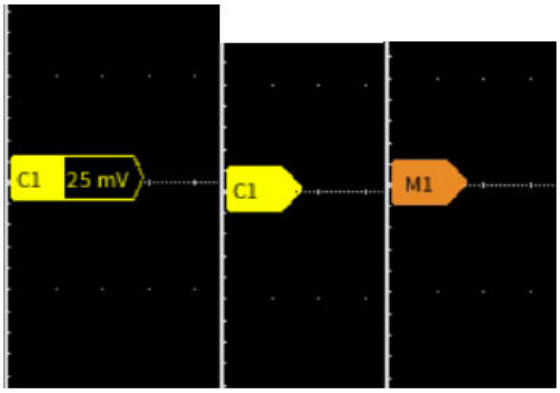
   When in Zoom mode, the Waveform Record View is replaced with the Zoom Overview.

2. The Expansion Point icon on the waveform view shows the center point around which the waveform expands and compresses when changing horizontal settings.
3. The Trigger Position Indicator shows where the trigger event occurred in the waveform record. The trigger icon is displayed in the waveform slice that is the trigger source.
4. The Zoom icon switches zoom on and off. The front panel Multipurpose knobs change the position and horizontal size of the Zoom Box.
5. The Trigger Level Indicator icon shows the trigger level on the trigger source waveform. Some trigger types require two trigger levels.
6. Measurement badges show measurement results.
7. The Results Bar Handle opens or closes the results bar, to maximize waveform screen viewing when needed. To reopen the results bar, either tap the handle icon or swipe left from the right side of the display.

8. The System badges show global instrument settings (**Horizontal**, **Trigger**, Run/Stop status, and Date/Time).
9. The Inactive Channel buttons add channel waveforms to the Waveform view and add an associated Channel badge to the Settings bar.

   The optional **AFG** button opens the AFG configuration menu to set and enable the AFG output. This button is only present in S and OL model instruments..
10. Double-tap a badge to open its associated configuration menu. If you add more Channel or Waveform badges than can fit in the waveform badge display area, tap the scroll buttons at each end of the waveform badge area to scroll and display hidden badges.
11. The Waveform Handles on each waveform identify the source of that waveform (C<n> for channels, M<n> for Math waveforms, R<n> for Reference waveforms, B<n> for bus waveforms). The waveform handles are at the zero-volt level of the waveform by default. If an offset is present, then the handle shows the offset next to the source. The currently selected waveform handle is a solid color; unselected waveform handles are outlined.

    Double-tapping a waveform handle opens the configuration menu for that waveform.

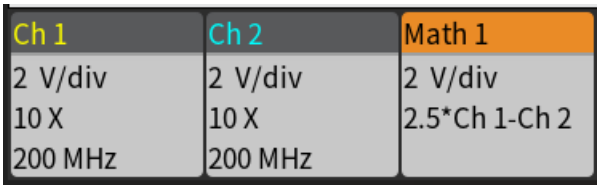


Badges

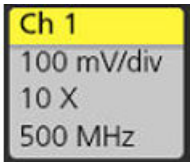
Badges are rectangular icons that show waveform, measurement, and instrument settings or readouts. Badges also provide fast access to configuration menus. The badge types are Channel, Waveform, Measurement, and System.

Channel and Waveform badges

Channel and Waveform (**Math**, **Ref**, **Bus)** badges are shown in the settings bar, located along the bottom left of the screen. Each waveform has its own badge. The badges show high-level settings for each displayed channel or waveform. Double-tap a badge to open its configuration menu.



Most Channel and Waveform badges also have Scale buttons, shown by single-tapping the badge. Use the Scale buttons to increase or decrease the vertical scale setting for that waveform.



You can drag Channel and Waveform badges to change their position in the **Settings** bar and open the badge right-click menu to access a quick-action menu.

There are two ways to delete Channel and Waveform badges.

- Right-click the badge and turn it off.
- Flick the badge off the bottom edge of the display to remove it from the **Settings** bar.

Channel badges are listed in the channel order. Channel badges may also display short error or warning messages. For more information double-tap the badge to open its configuration menu, or search the instrument Help.

Waveform badges (**Math**, **Ref**, **Bus)** are listed in the order created, and are grouped together by type. Deleting a Waveform badge does not change the order or names of the remaining badges.

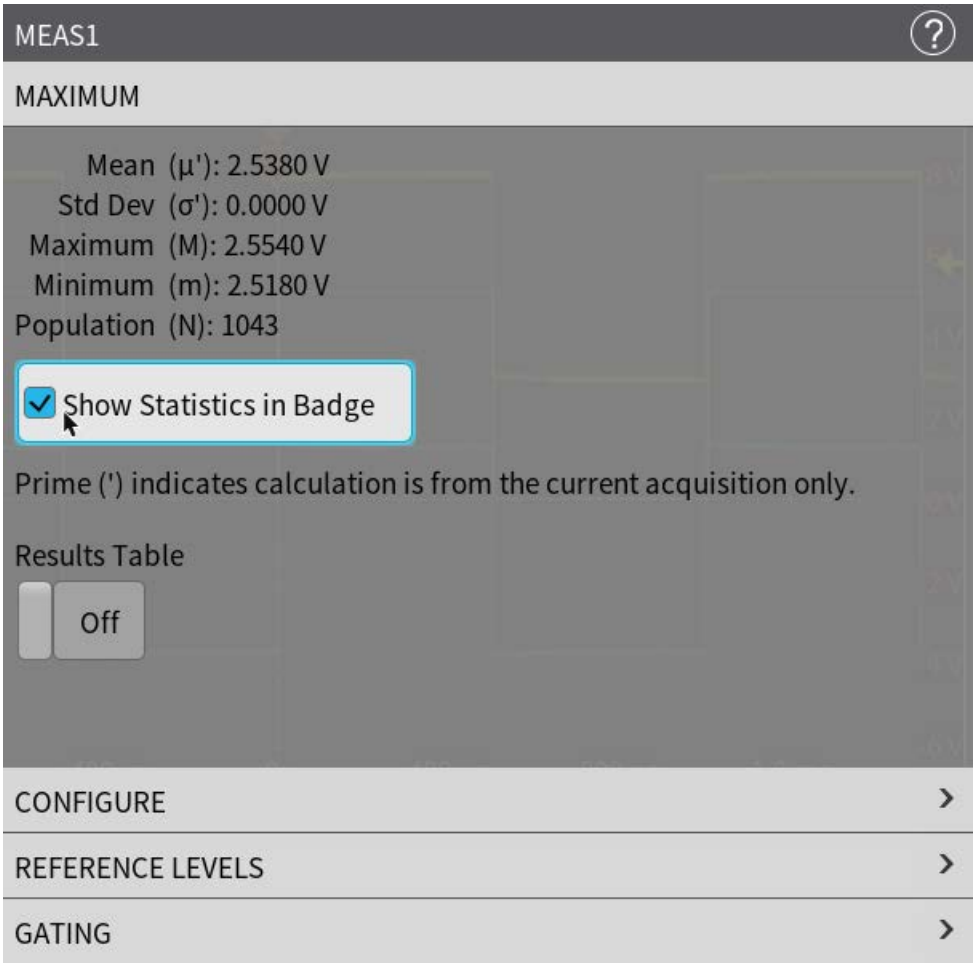
Measurement badges

Measurement badges are located in the **Results** bar. They show measurements or search results. The badge title also shows the measurement source or sources. To add a Measurement badge, tap the **Measure** button and select a measurement.

Double-tap a Measurement badge to open its configuration menu to change or refine settings. The default measurement badge readout shows the measurement's mean (μ) value.

Wide badge 1 series

To add statistical readouts to individual measurement badges, double-tap a measurement badge to open its configuration menu and select **Show Statistics in Badge**. The measurement badge displays the standard deviation (σ) value. The standard deviation is zero, when the population is one.



Measurement badges are listed in the order created, starting at the top of the Results bar. Deleting a Measurement badge does not change the order or names of the remaining badges.

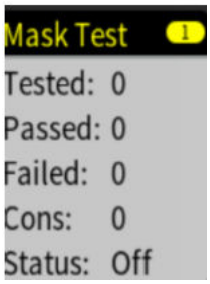
Right click on the badge to access a quick-action menu.

There are two ways to delete Channel and Waveform badges.

- Right-click the badge and turn it off.
- Flick the badge off the right edge of the display to remove it from the **Results** bar.

Mask Test badge

The mask test results and measurement statistics are displayed in the **Mask Test** badge in the Results bar. The badge is created when the Mask button is pressed and the mask measurement is added.



Badge readout	Description
Label (optional readout)	A label defined in the badge configuration menu.
Tested	The total number of waveforms tested against the mask.
Passed	The number of waveforms that contained no samples that violated the mask.
Failed	The number of waveforms that contained one or more samples that violated the mask. Displays in red if greater than or equal to the total failures threshold.
Cons	The highest number of consecutively failed waveforms in the test run. Displays in red if greater than or equal to the consecutive failures threshold.
Status	The status of the mask test. It can be On, Off, Passed/Passing (green) or Failed/Failing (red).

Double-tap a Mask Test badge to open its configuration menu to change or refine settings.

Right click on the badge to access a quick-action menu.

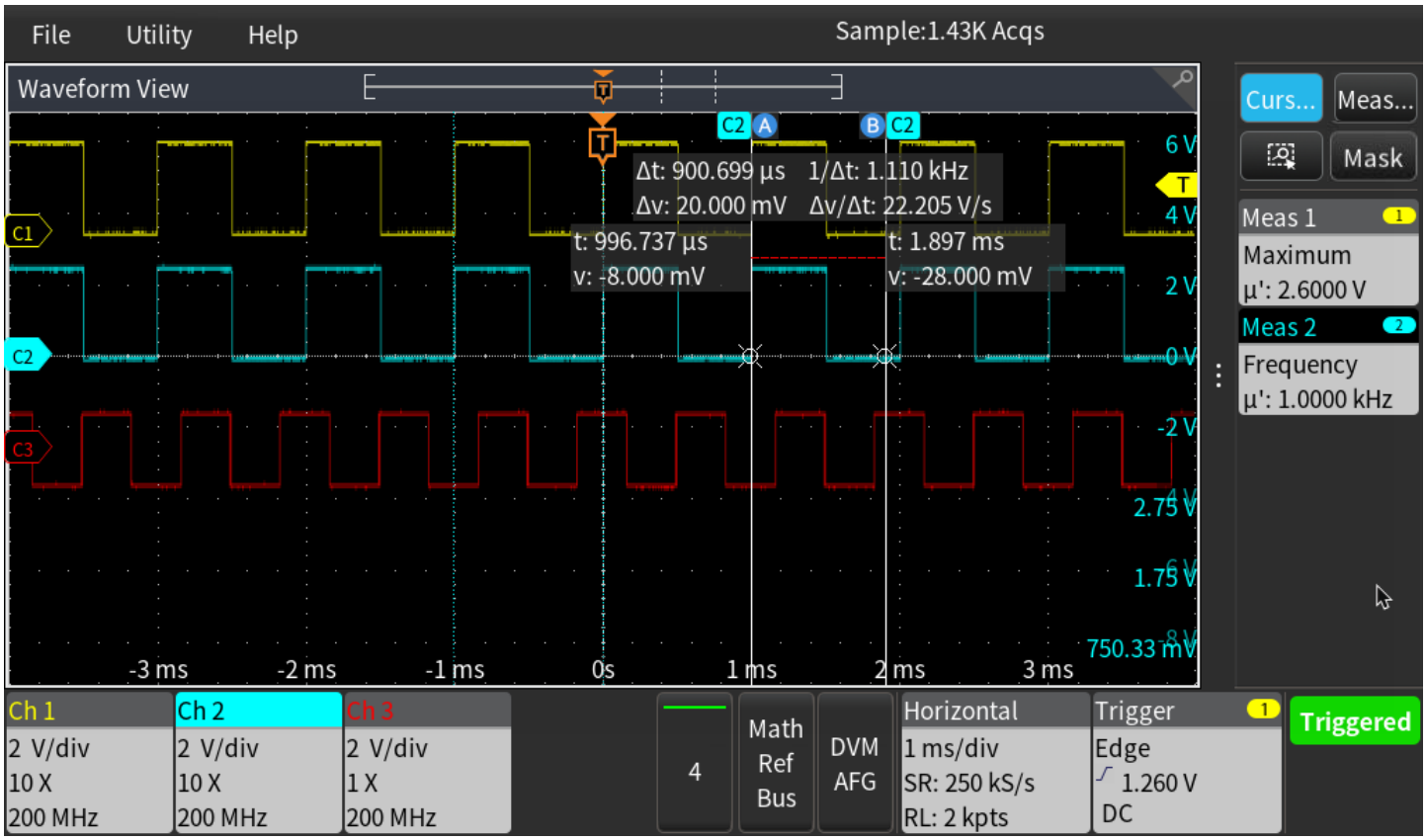
There are two ways to delete Channel and Waveform badges.

- Right-click the badge and turn it off.
- Flick the badge off the right edge of the display to remove it from the **Results** bar.

See for the mask test configuration settings Create a Mask on Waveforms (on page 59) for more information about masks and the Mask Test badge configuration menu (on page 97).

Cursor badges

You can display the cursor readouts in a **Cursors** badge in the Results bar. The badge contents depend on the cursor in use.



To create a cursor readouts badge, turn on **Cursors**, double-tap a cursor readout to open its configuration menu, and set the **Readouts** mode to **Badge**.

NOTE: You can only view cursor readouts in one location at a time; either on the waveform or in a Cursors badge.

You can drag the badge to change its position in the **Results** bar and open the badge right-click menu to access a quick-action menu.

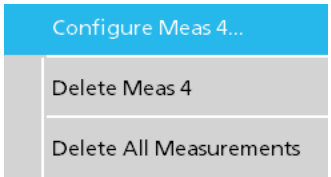
There are two ways to delete a Cursor badge:

1. Flick the badge off the right edge of the display to remove it from the **Results** bar.
2. Perform a long hold to bring up the quick action menu and turn it off.

Deleting mass Measurements badges at once

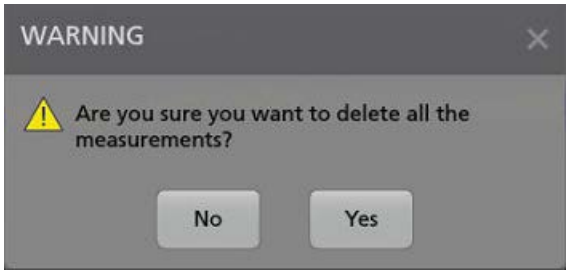
This badge helps you to delete/remove many number measurements that are in the Results bar.

1. Select and right click the Measurement badge in the **Results** bar, which displays the dialog box as shown:



Controls	Description
Configure Measurement	Configure Measurement badges
Delete Measurement	Deletes the selected Measurement badge in the Results bar.
Delete All Measurement	Deletes all the Measurement badges in the Results bar.

2. When **Delete All Measurements** is selected, the oscilloscope asks for the confirmation to delete all the measurements at a time.



Signal clipping and badges

WARNING: Clipping is caused by excessive or dangerous voltage at the probe tip, and/or a vertical scale setting that is not adequate to display the entire vertical range of the waveform. Excessive voltage at the probe tip can injure the operator and cause damage to the probe and/or instrument.

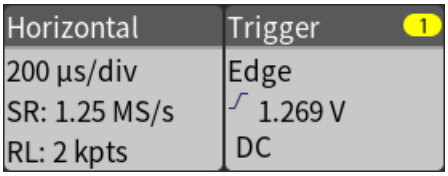
This instrument shows a warning triangle symbol and the words Clipping in a Channel badge when a vertical clipping condition exists. Any measurement badges associated with that channel also indicate a clipping condition by turning the measurement text red and listing the type of clipping (positive or negative).

To close the clipping message, change the vertical scale to show the entire waveform, disconnect the probe tip from the excessive voltage source, and check that you are probing the correct signal using the correct probe.

Clipping causes inaccurate amplitude-related measurement results. Clipping also causes inaccurate amplitude values in saved waveform files. If a math waveform is clipped, it will not affect amplitude measurements on that math waveform.

System badges

System badges (in the settings bar) display the main Horizontal and Trigger settings. You cannot delete System badges.



Double-tap a System badge to open its configuration menu.

The Horizontal badge also has Scale buttons, shown by single-tapping the badge. Use the Horizontal Scale buttons to increase or decrease the horizontal time settings.

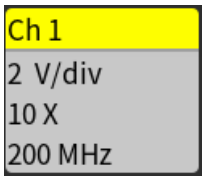
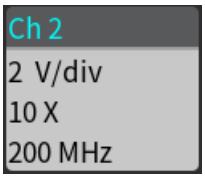
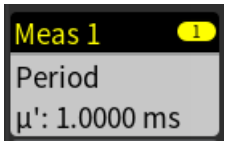
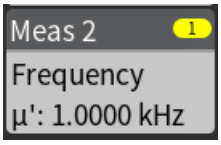
Common badge actions

Action	Result	Example
Single tap	Immediate access controls (Scale)	

Action	Result	Example
Double tap	Configuration menu with access to all settings for the badge.	
Touch and hold	Right-click menu with single tap access to common actions. Typical actions include turning off a channel and deleting a measurement or search badge.	
Flick	Flick the badge off the bottom edge of the display to remove it from the **Settings** bar. Flick the badge off the right edge of the display to remove it from the **Results** bar.	

Badge selection status

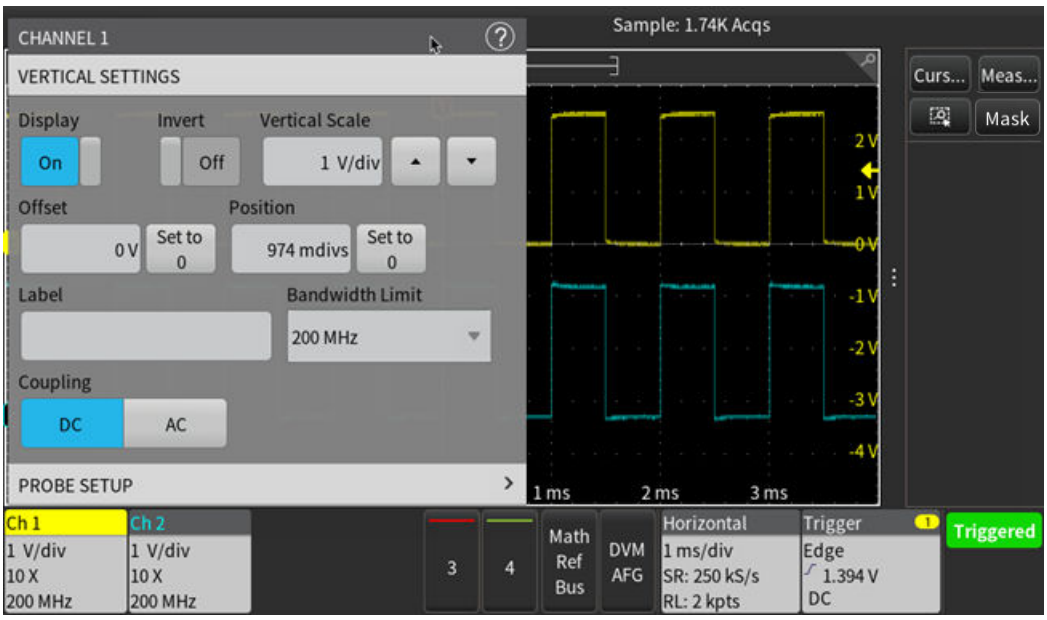
The appearance of a badge indicates its selection status (selected or unselected), or if a measurement needs to be deleted to close a channel or waveform badge.

Badge type	Selected	Unselected
Channel or Waveform		
Measurement		

Configuration menus

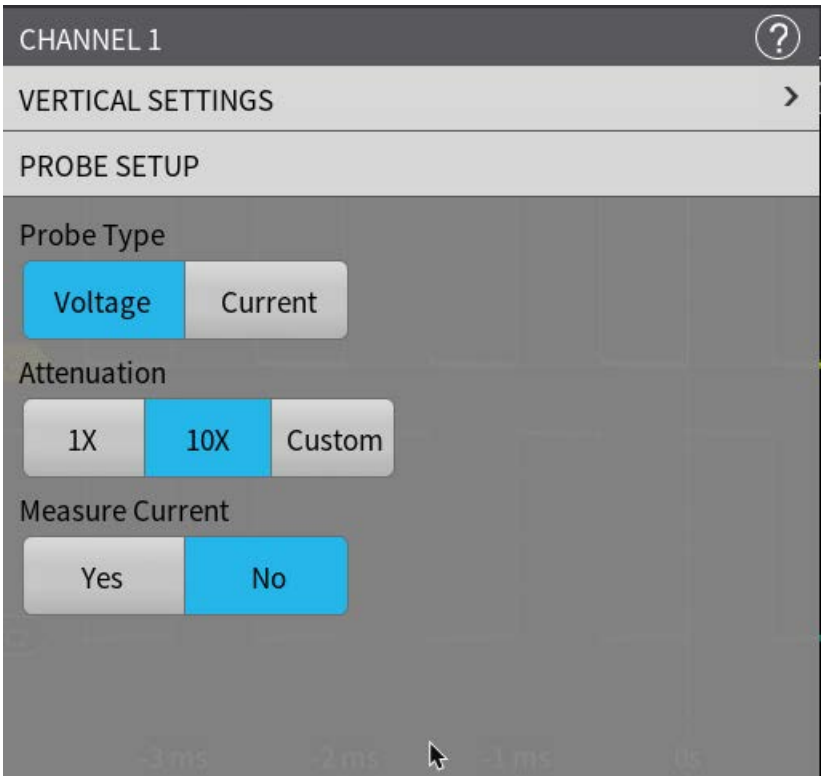
Configuration menus let you quickly set the parameters for channels, system settings (Horizontal, Trigger, measurements, cursor readouts, Waveform and Plot views, and so on.

Double-tap an item (badge, **Waveform View** or **Plot View**, cursor readouts, and so on) to open its configuration menu. For example, double-tap a Channel badge in the **Settings Bar** to open its configuration menu.



Selections or values that you enter take effect immediately. Menu contents are dynamic, and can change depending on your selections, instrument options, or attached probes.

Related settings are grouped into 'panels'. Tap the panel name to show those settings. Changes to panel settings can change the values and/or fields shown in that panel and other panels.



Tap anywhere outside a configuration menu to close it.

To open Help content for a configuration menu, tap the question mark icon in the upper right corner of the menu.

Zoom user interface

Use the zoom tools to magnify waveforms to view signal details.

1. The **Zoom Overview** shows the entire waveform record. All waveforms are shown in Overlay mode in the Zoom Overview area. Using pinch and expand gestures on the Zoom Overview waveforms changes the horizontal time base settings.
2. The **Zoom Box** shows the area of the Zoom Overview to display in the Zoom View (see 5). You can touch and drag the box to move the area to view. Moving the Zoom Box, or changing its position, does not change the horizontal time base settings.
3. The **Zoom** icon (in the upper right corner of the Waveform View) switches zoom mode on and off.
4. A zoom box lets you quickly draw a box around an area of interest in the Waveform or Zoom Overview.
5. The **Zoom View** shows the zoomed waveforms, as marked by the Zoom Box, in the Zoom Waveform Record View. Use pinch and/or drag options in the zoom view to change the zoomed area of interest. Pinch, expand, and drag gestures in the Zoom View only change zoom magnification settings and Zoom Box position.
6. Use the **Zoom Title Bar** controls to adjust the vertical and horizontal size of the zoom area. Click or tap the + or - buttons or use the A and B multipurpose knobs.



Double tap the **Horizontal Zoom Position** or **Horizontal Zoom Scale** fields to enter a value using a numerical keypad.

To exit the zoom display mode, tap the Zoom icon in the corner of display or tap the X in the Zoom Title Bar.

Using the touch screen interface for common tasks

Use standard touch screen actions, similar to those found on smart phones and tablets, to interact with most screen objects. You can also use a mouse to interact with the UI. The equivalent mouse operation is shown for each touch operation.

Task	Touchscreen UI action	Mouse action
Add a channel, math, reference, or bus waveform to the screen.	Tap an inactive channel button, or **Math Ref Bus** button (select the desired waveform from the secondary menu that opens) button.	Click an inactive channel button, or **Math Ref Bus** button (select the desired waveform from the secondary menu that opens) button.
Select a channel, math, reference, or bus waveform to make it active	Tap the Channel or Waveform badge or handle.	Left-click the Channel or Waveform badge or handle.
Display scale buttons on a badge (waveform, measurement, horizontal).	Tap the badge.	Click the badge.
Open a configuration menu on any item (all badges, views, cursor readouts, labels, and so on).	Double-tap the badge, view, or other object.	Double-click the badge, view, or other object.
Open a right-click menu (badges, views).	Touch and hold on the badge, Waveform View, Plot view, or other screen item until a menu opens.	Right-click the object.
Close a configuration menu. Some dialog boxes will not close until you click an OK, Close, or other button in the dialog.	Tap anywhere outside the menu or dialog.	Click anywhere outside the menu or dialog.

Task	Touchscreen UI action	Mouse action
Move a menu.	Touch and hold the menu title bar or a blank area in the menu, then drag the menu to new position.	Click and hold the right mouse button on title or blank area, drag to new position.
Change horizontal or vertical settings directly on a waveform. Vertical changes only apply to the selected channel or waveform; horizontal changes apply to all channels and waveforms.	Tap a badge and use the Scale buttons.	Left-click a channel, waveform, or **Horizontal** badge and click on the Scale buttons.
	Touch and hold two fingertips on the waveform view, move them together or apart vertically or horizontally, lift from screen; repeat.	
Increase or decrease the zoom area while in Zoom mode.	Touch and hold two fingertips on the waveform view, move them together or apart vertically or horizontally, lift from screen; repeat.	Click the **+** or **-** buttons on the Zoom Title bar.
		Click the **Draw-a-Box** button, draw a box around the waveform area of interest.
Quickly scroll or pan a waveform or list.	Touch and drag in the waveform or list.	Click and drag in the waveform or list.
Close or open the **Results Bar** to increase the **Waveform View** area.	Tap on the **Results Bar Handle** (three vertical dots in border) or anywhere in the border between the **Waveform View** and the **Results Bar**.	Click the **Results Bar Handle** (three vertical dots in border) or anywhere in the border between the **Waveform View** and the **Results Bar**.
		Click and drag the Results Bar divider.
Change the position of badges in the **Settings Bar** or **Results Bar**.	Touch and drag the badge to a new position in the same bar.	Click and drag the badge to a new position in the same bar.

Accessing application help

Use the instrument Help to quickly get information about a function or assistance in performing a task.

On the instrument: Tap **Help > Help** to access a PDF of the instrument Help.

On tek.com: Scan the QR code to access the most recent version on the tek.com website from your personal device.

Configure the instrument

Configurations to help operate your instrument efficiently.

Download and install the latest instrument firmware

Installing the latest firmware helps ensure that your instrument has the latest features and is taking the most accurate measurements.

Before you begin

Save any important on-instrument files such as waveforms, screen captures, and setups to a USB drive or network. The installation process does not remove user-created files, but it is recommended to back up important files before an update.

Determine the current version of firmware installed on the instrument using the **Help > About** menu.

Procedure

To download the instrument firmware and install onto an instrument:

1. Open up a Web browser on a PC and go to http://www.tek.com/product-support
2. Enter the instrument model number in the search field and click **Go**.
3. Scroll down the screen and click the **Software** tab.
4. If the listed available firmware version is newer than what is on your instrument, select and download that file to your PC.
5. Copy the downloaded firmware file to a USB drive.
6. Insert the USB drive with the firmware file into one of the USB ports on the instrument.
7. Power on the instrument.
8. Follow on-screen instructions.

   The instrument will take a few minutes to install the new firmware. Do not remove the USB flash drive or power off the instrument during this time.

What to do next

To confirm the firmware update, locate the version number found in the **About** window under the **Help** menu. Confirm that the instrument firmware version number matches the firmware version number that you just installed.

Set the time zone and clock readout format

Set the time zone to your region so that saved files are marked with the correct date and time information. You can also set the time format (12 or 24 hour clock).

Procedure

1. Double-tap the **Date/Time** badge (bottom-right of screen) to open the configuration menu.
2. To turn off showing the date and time on the screen, tap **Display** button to **Off**.

To turn on date/time display again, double-tap in the blank area where the date/time badge was displayed to open the configuration menu, and set the **Display** button to **On**.

3. Select a time format (**12 Hour** or **24 Hour**).
4. Tap the **UTC Offset** field and select the offset that applies to your location.
5. Tap anywhere outside of the menu to close it.

Functional check

Use this procedure to quickly verify that the oscilloscope can display a waveform and take a measurement.

1. Power on the oscilloscope.
2. Tap **Utility** > **Self test**. Check that all tests listed show **Pass**.
3. Connect an analog probe to the Channel 1 connector.
4. Connect the oscilloscope tip and ground lead to the probe compensation connectors. Refer to your specific probe's manual for setup instructions.
5. Push the **Autoset** button. You should see a square wave in the display. The levels should be approximately 0 V - 2.5 V and 1 kHz.
6. Tap the **Measure** button.
7. Tap the **Time Measurements** panel in the Add Measurements configuration menu.
8. Double-tap the **Frequency** button to add the frequency measurement to the Results bar.
9. Check that the **Frequency** measurement reads 1 kHz.
10. Repeat these steps to check the other channels on the oscilloscope. Make sure that you set the source in the Add Measurement configuration menu to use the correct channel before adding the Frequency measurement.

Run Signal Path Compensation (SPC)

Run SPC when you first receive an instrument and at regular intervals for best measurement accuracy. You should run SPC whenever the ambient (room) temperature has changed by more than 5 °C (9 °F), or once a week if you use vertical scale settings of 5 mV per division or less.

About this task

Signal Path Compensation (SPC) corrects for DC level inaccuracies in the internal signal path, caused by temperature variations and/or long-term signal path drift. Failure to run SPC on a regular basis may result in the instrument not meeting warranted performance levels at low volts per division settings.

Before you begin

Disconnect all probes and cables from the front-panel channel inputs and rear-panel signal connectors.

Procedure

1. Power on and warm up the instrument for at least 20 minutes.
2. Tap **Utility > Calibration**.
3. Tap **Run SPC**. The **SPC Status** readout shows **Running** while SPC is running. SPC can take approximately 3 minutes per channel to run, so wait until the SPC Status message changes to **Pass** before reconnecting probes or cables and using the instrument.

   CAUTION: You can abort the SPC calibration by tapping **Abort SPC**. This may leave some channels uncompensated, resulting in possible inaccurate measurements. If you do abort the SPC, make sure to run the SPC procedure completely before using the instrument to take measurements.

4. Close the **Calibration** configuration dialog when SPC has completed.

If the SPC fails, write down any error message text. Make sure that all probes and cables are disconnected and run the SPC again. If the SPC still fails, contact Tektronix Customer Support.

Compensate the probe

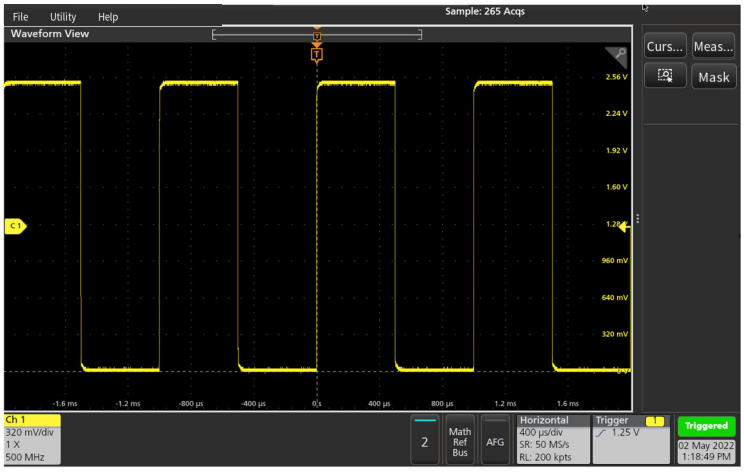
Probe compensation adjusts the high frequency response of a probe for best waveform capture and measurement accuracy. Use this procedure to adjust probe compensation for probes with a manual adjustment.

NOTE: You must run this procedure every time you move a probe from one channel to another.

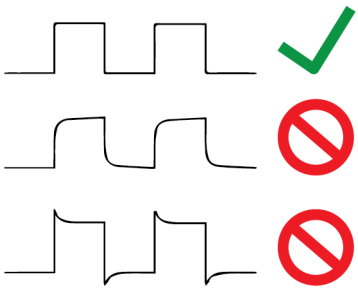
Due to slight differences in front end architecture, probes should be recompensated when changing the vertical scale between 50 mV/div (1X) and 55 mV/div (1X).

The following procedure checks the probe compensation.

1. Connect a supported probe to Channel 1.
2. Connect the probe tip to the probe compensation terminal labeled **~2.5 V**. Connect the ground lead to the probe compensation terminal labeled **PROBE COMP**. Connect only one probe at a time to the **PROBE COMP** terminals.
3. Turn on the Channel 1 and turn off all other channels.
4. Tap **File** > **Default Setup**.
5. Push the **Autoset** button on the front-panel or tap **File** > **Autoset** from the menu bar. The screen should display a square wave with the levels between approximately 0 V - 2.5 V and 1 kHz.



6. Check the shape of the displayed waveform to determine if the probe needs adjusted. If the waveform has a square leading edge and a flat top and bottom, the probe does not need adjusted. If the waveform leading edge is rounded or has a spike, you need to adjust the probe compensation.



7. Use the probe adjustment tool provided with the probe to adjust the probe until the waveform has as flat a top and bottom as possible. Remove the adjustment tool before looking at the waveform. Repeat until waveform top and bottom are flat. See your probe manual for adjustment location and instructions.
8. Repeat these steps for each probe connected to each channel.

Connect to a network (LAN)

Connecting to a network allows you to remotely access the instrument.

Work with your network administrator to obtain the required information to connect to your network (IP address, Gateway IP address, Subnet Mask, DNS IP address, and so on).

1. Connect a CAT5 cable from the instrument LAN connector to your network.
2. Select **Utility > I/O** on the menu bar to open the I/O configuration menu.
3. Obtain or enter the network address information:
   - If your network is DHCP-enabled, and the IP address field does not already show an address, tap **Auto** to obtain the IP address information from the network. DHCP mode is the default mode.
   - If your network is not DHCP-enabled, or you need a permanent (non-changing) IP address for this instrument, tap **Manual** and enter the IP address and other values provided by your IT or system administrator resource.
4. Tap **Test Connection** to verify that the network connection is working. The **LAN Status** icon turns green when the instrument successfully connects to your network. If you have problems connecting to your network, contact your system administration resource for help.

Connect the oscilloscope to a PC using a USB cable

Use a USB cable to connect the oscilloscope directly to a PC for remote instrument control.

1. On the oscilloscope, select **Utility > I/O** from the menu bar.
2. Tap **USB Device Port Settings**.
3. Confirm that the USB Device Port control is **On** (default setting).
4. Connect a USB cable from the PC to the USB **Device** port on the instrument.
5. Use the USB connection to remotely control the oscilloscope using PI commands.

Connect a mouse

The instrument supports most standard USB-connected and wireless mice (using a USB-connected dongle).

Connect a mouse by connecting the USB cable, or USB dongle, into the available USB Host port. The mouse should work immediately. If it does not, try the following:

1. Remove and reinsert the USB cable or dongle in the same port.

ESD Prevention Guidelines

Electrostatic discharge (ESD) can damage oscilloscope and some probe inputs. This topic discusses how to avoid this type of damage.

Electrostatic discharge (ESD) is a concern when handling any electronic equipment. The instrument is designed with robust ESD protection, however it is still possible that large discharges of static electricity directly into the signal input may damage the instrument. Use the following techniques to prevent electrostatic discharge from damaging the instrument.

- Discharge the static voltage from your body by wearing a grounded antistatic wrist strap while connecting and disconnecting cables, probes, and adapters. The instrument provides a ground connection to which to attach a wrist strap (near the Probe Comp ground connector).
- A cable that is left unconnected on a bench can develop a large static charge. Discharge the static voltage from all cables before connecting them to the instrument or device under test by momentarily grounding the center conductor of the cable, or by connecting a 50 Ω termination to one end, before attaching the cable to the instrument.
- Before you apply power, connect the instrument to an electrically-neutral reference point, such as earth ground. To do this, plug the three-pronged power cord into an outlet grounded to earth ground. Grounding the oscilloscope is necessary to ensure safety and to take accurate measurements.
- If you are working with static sensitive components, ground yourself. Static electricity that builds up on your body can damage static-sensitive components. Wear a wrist strap to safely send static charges on your body to earth ground.
- The oscilloscope must share the same ground as any circuits that you plan to test.

Analog channel operating basics

Acquiring a signal

After acquiring a signal you can take measurements and plot the results.

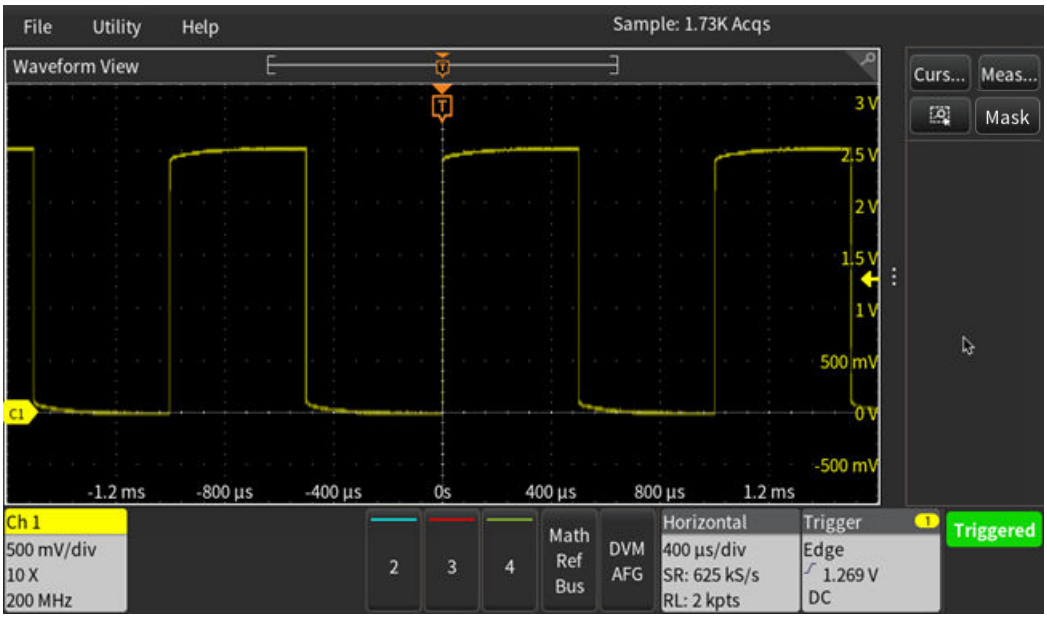
Use the following procedure to set the scale and position parameters for analog signal acquisition.

1. Press the **Default Setup** button.
2. Connect the probe output to the desired oscilloscope channel, and connect the probe input to the input signal source using proper probing/connecting techniques.

   NOTE: Some probes automatically set their termination and other values.

3. Tap the channel button to add the channel waveform to the waveform view and add a channel badge to the settings bar. A channel button lights when its channel is on.
4. Double-tap the channel badge to open the channel Vertical Settings menu. To change the input coupling, select the appropriate coupling button.
   - Select DC to couple both the AC and DC components of an input signal.
   - Select AC to couple only the AC components of an input signal.
5. Use the Vertical knobs to scale and position the waveform vertically on the screen. The knobs should be highlighted with the active channel's color. Dragging the waveform handle also positions the waveform.
6. Use the Vertical Settings menu to change the offset. Tap **Offset** and then use a multipurpose knobs to adjust the offset.
7. Use the Horizontal knobs to scale and position the waveform horizontally on the screen and to set the record length. Dragging the reference icon also positions the waveform.
8. Use the Horizontal menu to set the record length.
9. You may attempt to stabilize the display by pressing the trigger Level knobs to set the trigger level to 50%. The 50% level is calculated as the midpoint between the highest and lowest samples of the acquired waveform. If your signal is periodic you should see a stable, triggered signal. This method will not work as well with random signals.

Autoset to quickly display a waveform



The Autoset function analyzes the signal characteristics and changes the instrument Horizontal, Vertical, and Trigger settings to automatically display a triggered waveform. You can then make further changes to trigger and horizontal settings to view the waveform point of interest.

1. Connect the probe with the signal of interest to an available channel.
2. Double-tap the **Trigger** badge and set the trigger source to that of the signal of interest.
3. Connect any other associated signal(s) to available channel input(s).
4. Add the channel waveforms to the Waveform view. See Add a channel waveform to the display (on page 40).
5. Tap **File** > **Autoset** or push the front-panel **Autoset** button.

The instrument adjusts the horizontal and trigger settings of the trigger source channel to display a triggered waveform for that channel. Vertical scale and position adjustments for all active channels are controlled by the **Autoset in Overlay Display Mode Optimizes** selection in the **Autoset** panel of the **User Preferences** menu. If the selection is **Visibility**, Autoset vertically scales and positions all active channel waveforms such that they are uniformly spaced on screen. If the selection is **Resolution**, Autoset vertically scales and positions all active channel waveforms such that they each use as much of the ADC's range as possible.

NOTE: You can set which parameters the instrument can adjust when running an Autoset. Access the Autoset panel in **Utility > User Preferences > Autoset.** See User Preferences (Utility menu).

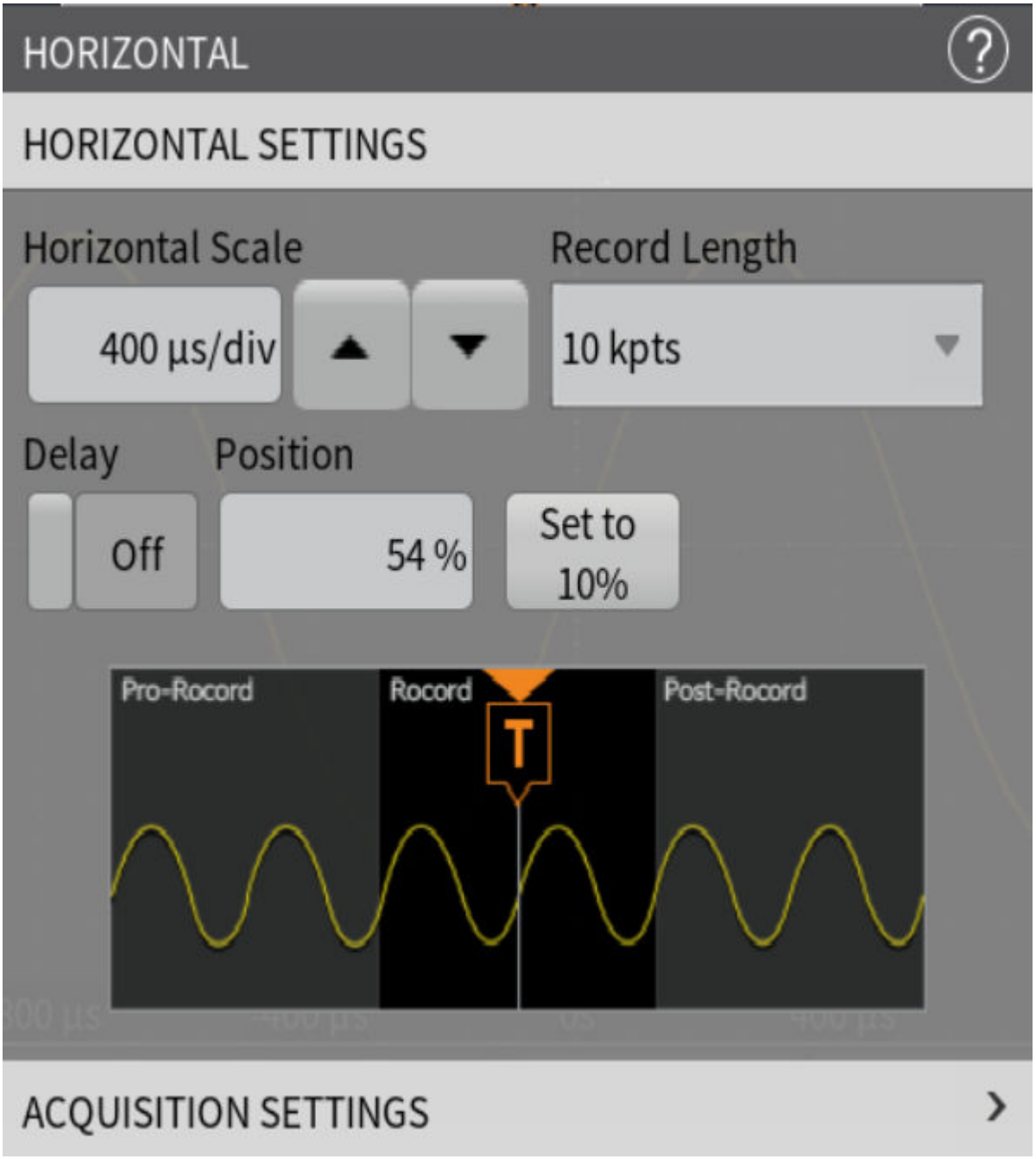
Autoset guidelines

- Autoset displays three or four cycles (depending on the detected signal) with the trigger level near the midlevel of the signal.
- The trigger is set to type Edge, rising slope, DC coupling.
- If no channels are displayed before pushing **Autoset**, the oscilloscope adds Ch 1 to the Waveform view whether it has a signal or not.
- Autoset ignores math, reference, and bus waveforms.
- A channel or waveform with a frequency less than 40 Hz is classified as no signal.

Set Horizontal parameters

Use this procedure to set the horizontal time base parameters such as mode, horizontal scale, delay, and trigger delay time (relative to the center of the waveform record).

1. Double-tap the **Horizontal** badge on the Settings bar to open the Horizontal configuration menu.

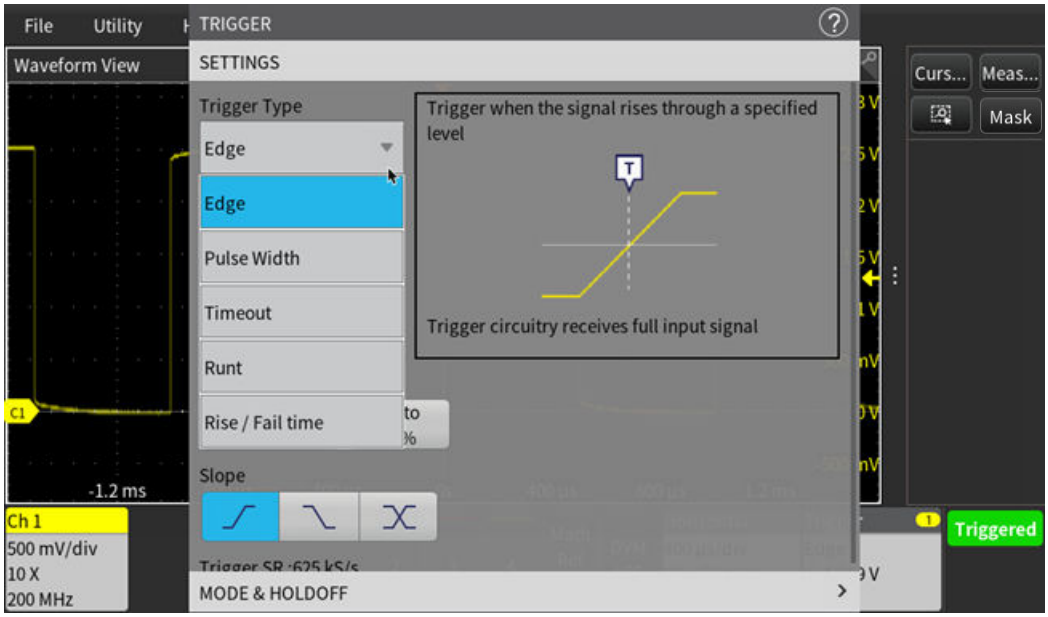


1. Use the menu selections to set horizontal parameters.
2. Tap the **Help** icon on the menu title for more information on these settings.

How to trigger on a signal

Use this procedure to open the Trigger menu to select and configure the trigger event type and conditions.

1. Double-tap the **Trigger** badge on the Settings bar to open the Trigger configuration menu.
2. Select a trigger from the **Trigger Type** list. The trigger type sets what fields are available in the menu and also updates the illustration to show a graphic of the trigger type.



   To trigger on a bus, you must first add the bus to the Waveform view. See Add a math, reference, or bus waveform (on page 41).
3. Select the other fields and panels to refine the trigger conditions. The menu fields and trigger graphic updates as you make changes to the trigger settings. Displayed fields depend on the selected trigger type. Selection changes take effect immediately.
4. Tap the **Help** icon on the menu title for more information on these settings.
5. Tap outside the menu to close the menu.

See also:

Advanced triggering

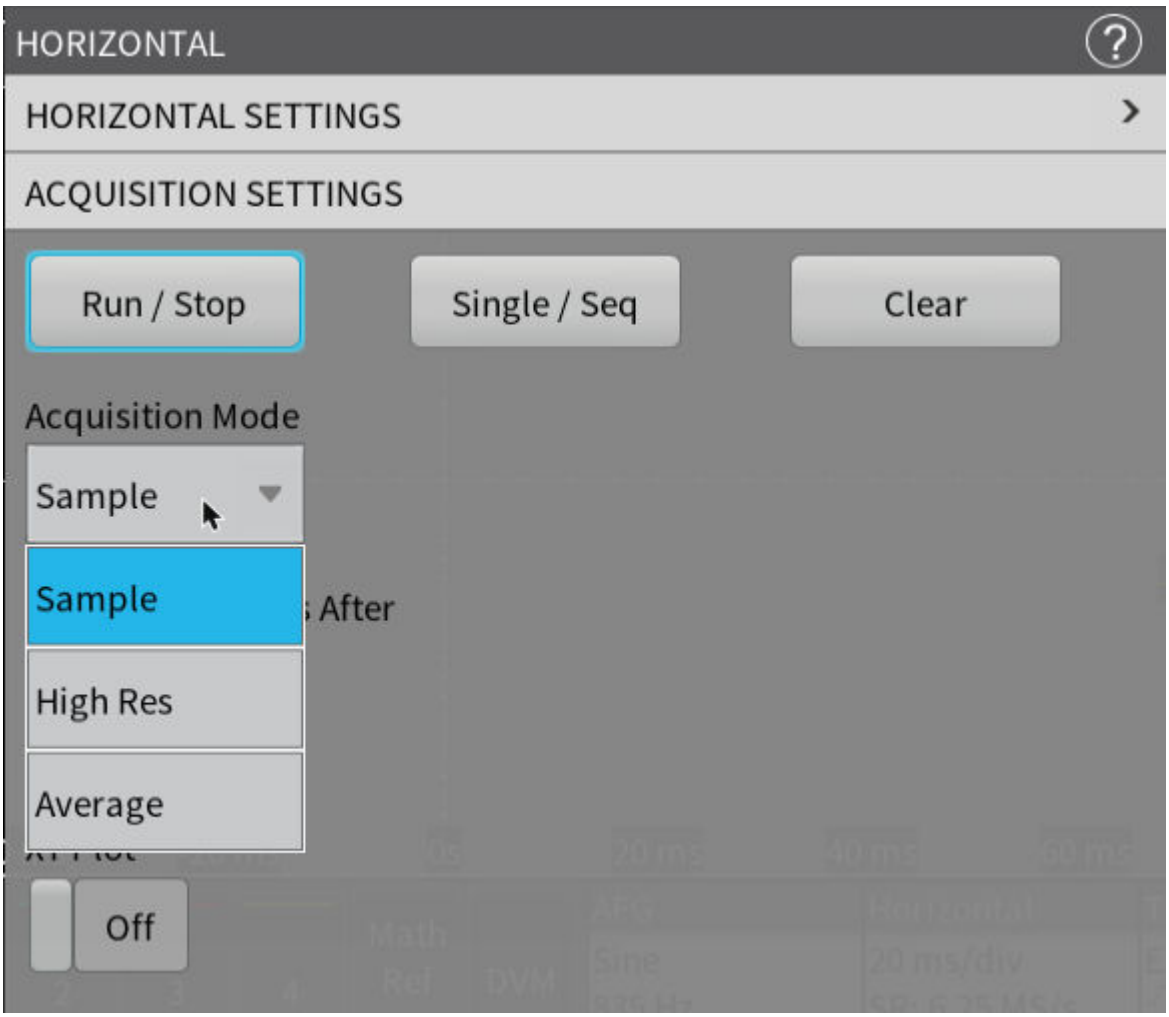
Trigger configuration overview (on page 108)

Triggering concepts (on page 122)

Set the acquisition mode

Use this procedure to set the method the instrument uses to acquire and display the signal.

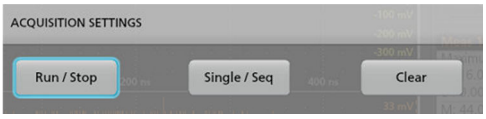
1. Double tap the Horizontal badge.
2. Tap **Acquisition Settings**.
3. Tap outside the menu to close the menu.



Start and stop an acquisition

Acquisition controls the start and stop of waveform acquisition.

1. To start an acquisition, double-tap the **Horizontal** configuration menu, tap the **Acquisition Settings** on the Settings bar. Tap **Run/Stop** in the Acquisition configuration menu.



2. Tap the **Stopped** button on the setting bar to run the acquisition again. You can also push the **Run/Stop** button on the front panel.
3. To stop an acquisition tap **Run/Stop** again or push the **Run/Stop** button. Tap the **Running** button on the setting bar.
4. To take a single acquisition:
   a. Double-tap the **Horizontal** configuration menu.
   b. Tap **Acquisition Settings** on the Settings bar.
5. Tap **Single/Seq** in the Acquisition configuration menu or push the **Single/Seq** button on the front panel.
6. The color of the **Run/Stop** and **Single/Seq** buttons on the front panel indicate the acquisition status (green = acquiring, red = stopped).
7. To clear the current acquisition data from waveform memory, double-tap the Acquisition badge and tap **Clear** in the Acquisition configuration menu, or push the **Clear** button on the front panel.

Add a channel waveform to the display

Use this procedure to add a channel signal to the Waveform View.

1. Connect signal(s) to the channel input(s).
2. Tap an Inactive Channel button (in the Settings bar) of a connected channel.

   The selected channel is added to the Waveform View and a Channel badge is added to the Settings bar.
3. Continue tapping Inactive Channel buttons to add more channels. The waveforms are displayed one on top of another.
4. Double-tap a channel badge to open that channel's configuration menu to check or change settings.

Configure channel or waveform settings

Use the channel and waveform configuration menus to set parameters such as vertical scale and offset, coupling, bandwidth, probe settings, deskew values, external attenuation values, and other settings.

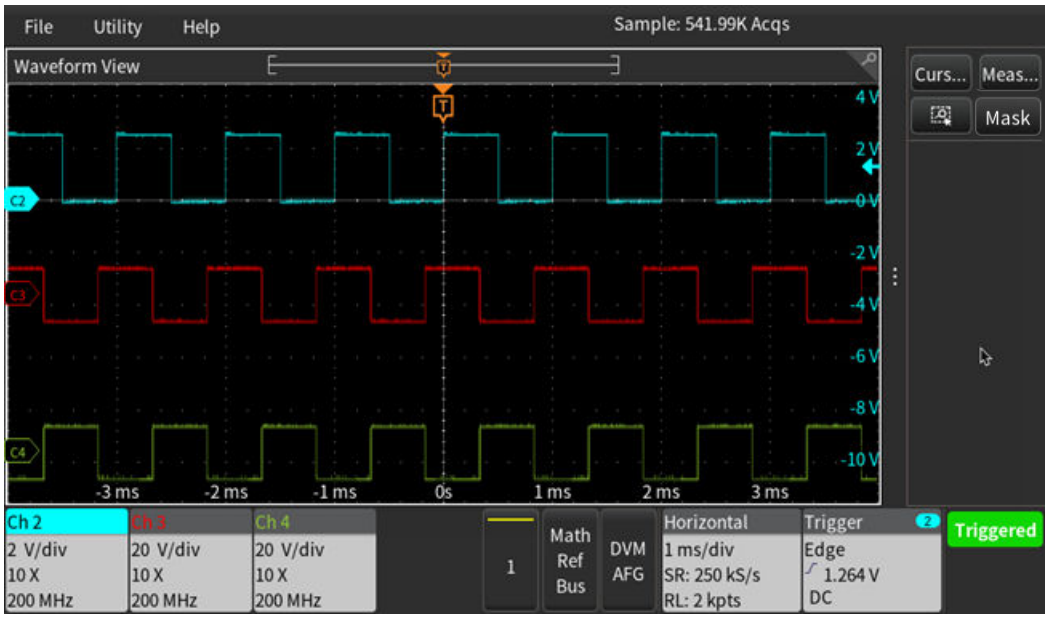
Before you begin

Prerequisite: There is a channel or waveform badge in the Settings bar.

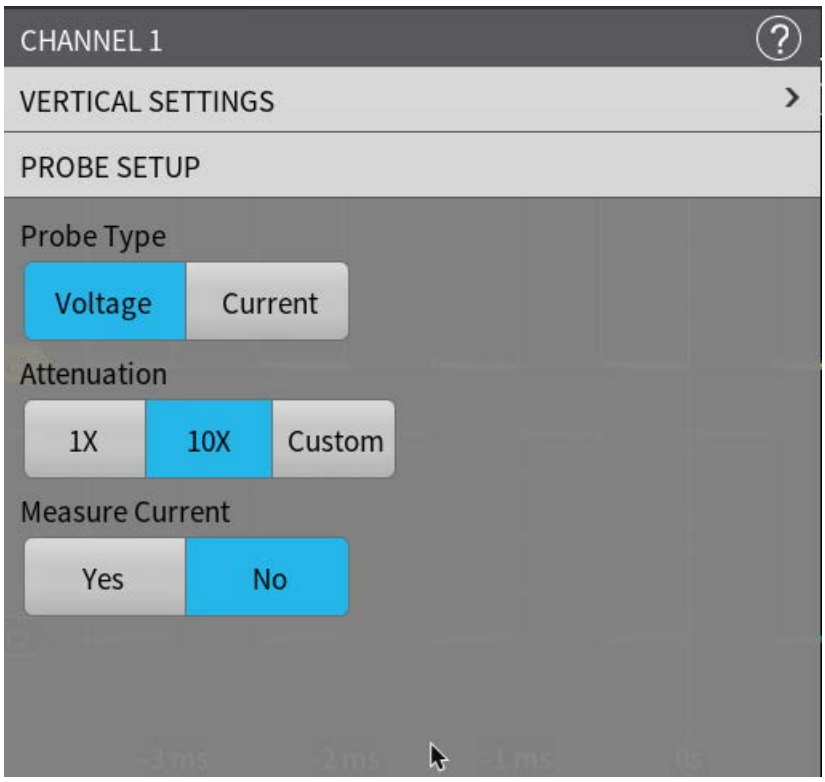
Procedure

1. Double-tap a **Channel** or **Waveform** badge to open a configuration menu for that item.

   For example, in a Channel menu, use the **Vertical Settings** panel to set basic channel parameters such as vertical scale and position, offset, coupling, termination, and bandwidth limit. Available settings depend on the waveform type.



2. Tap the **Probe Setup** panel to confirm probe settings.



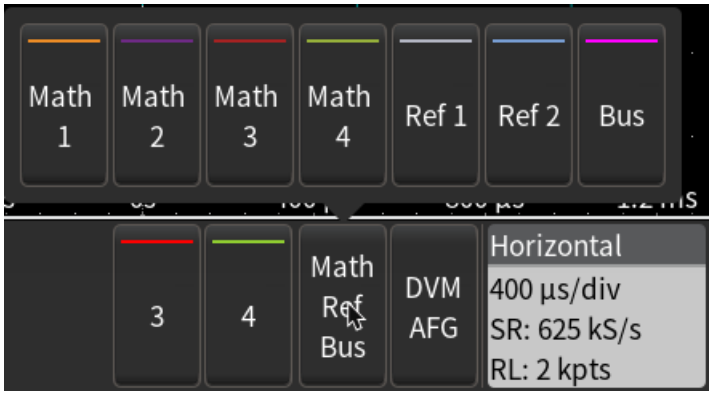
3. Tap the Help icon on the menu title to open the help topic for more information.
4. Tap outside the menu to close the menu.

Add a math, reference, or bus waveform

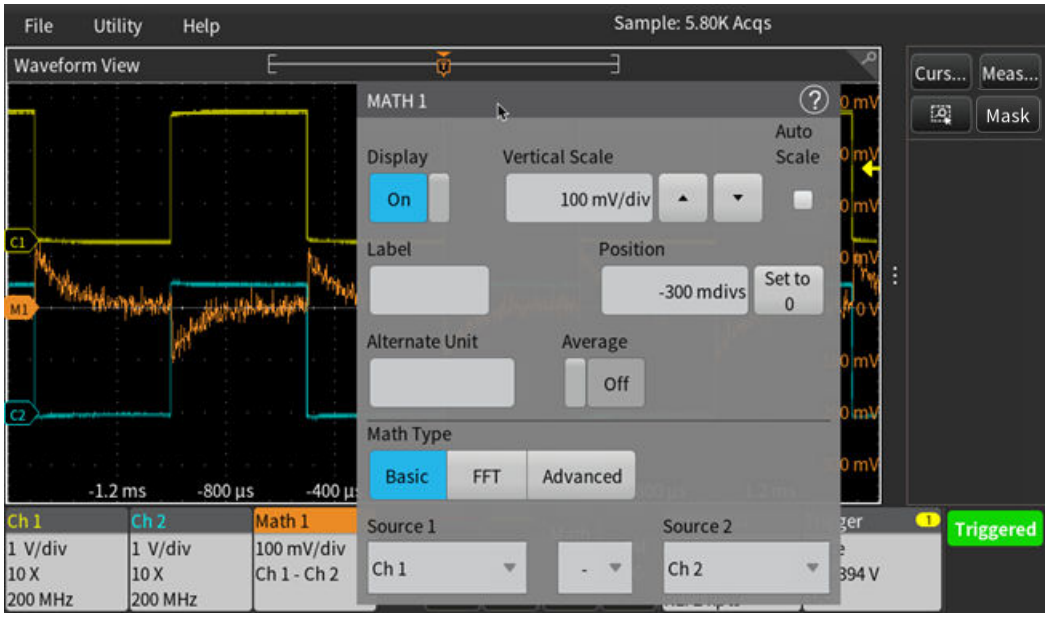
Math waveforms let you create new waveforms based on operations between two or more waveforms or by applying equations to waveform data. A reference waveform is a static waveform record displayed for comparison. Bus waveforms let you view and analyze serial or parallel data.

There is no set limit to the number of Math, Reference, or Bus waveforms you can add to the Waveform View, other than system physical memory constraints.

1. Tap the **Math Ref Bus** >**Add New Math**, **Add New Ref**, or **Add New Bus** button in the Settings bar.



2. The instrument adds the waveform to the Waveform view, adds a Waveform badge to the Settings bar, and opens a configuration menu. This example shows adding a Math waveform.



3. Use the configuration menus to refine the waveform parameters. Displayed fields depend on the waveform and selections made in the menu. Selection changes take effect immediately.

   This example shows adding a Math waveform, using the Math **Source** fields to select Ch 1 and Ch 2 as the waveform sources, set the math type to **Basic** math operation, and subtracting channel 2 from channel 1.

4. When adding a Reference waveform, the instrument displays a **Recall** configuration menu. Navigate to and select the reference waveform file (*.wfm) to recall, then tap the **Recall** button. The instrument displays the Reference waveform.

5. Double-tap a math, reference, or bus badge to check or change the waveform settings. See Configure channel or waveform settings (on page 40).

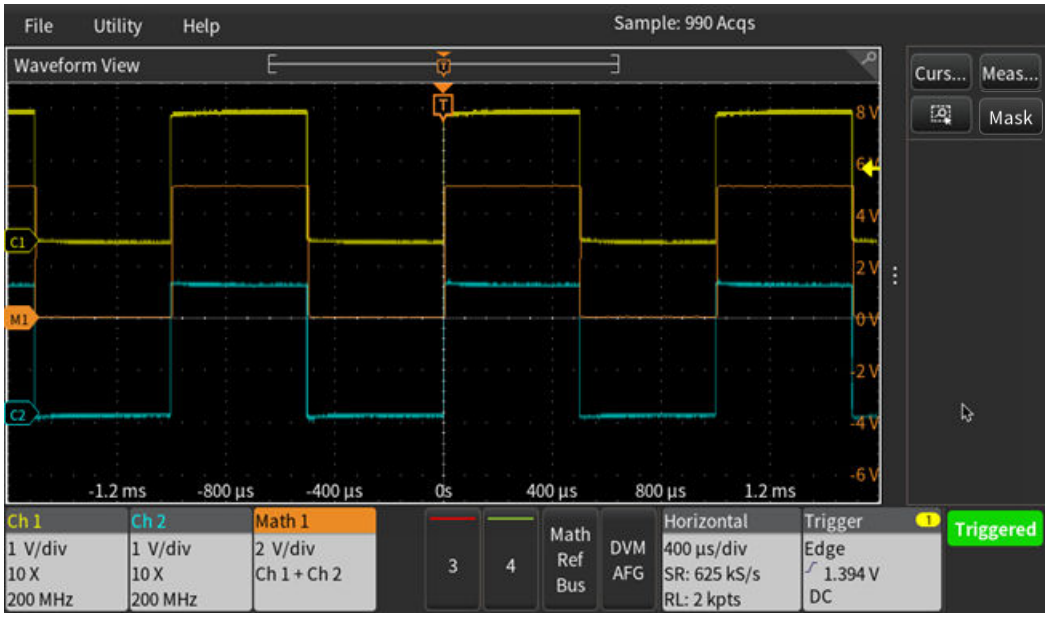
6. Tap the Help icon on a configuration menu title for more information on math, reference, and bus waveform settings.

7. Tap outside the menu to close the menu.

Add a measurement

Use this procedure to select and add measurements.

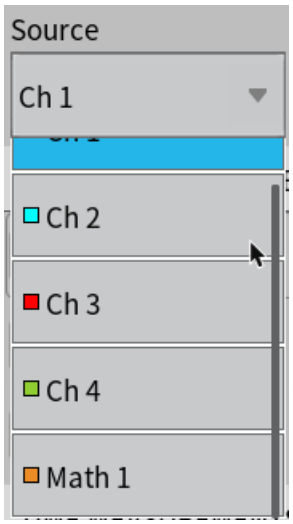
1. Acquire the channel(s) and/or waveform(s) on which you want to take measurements.



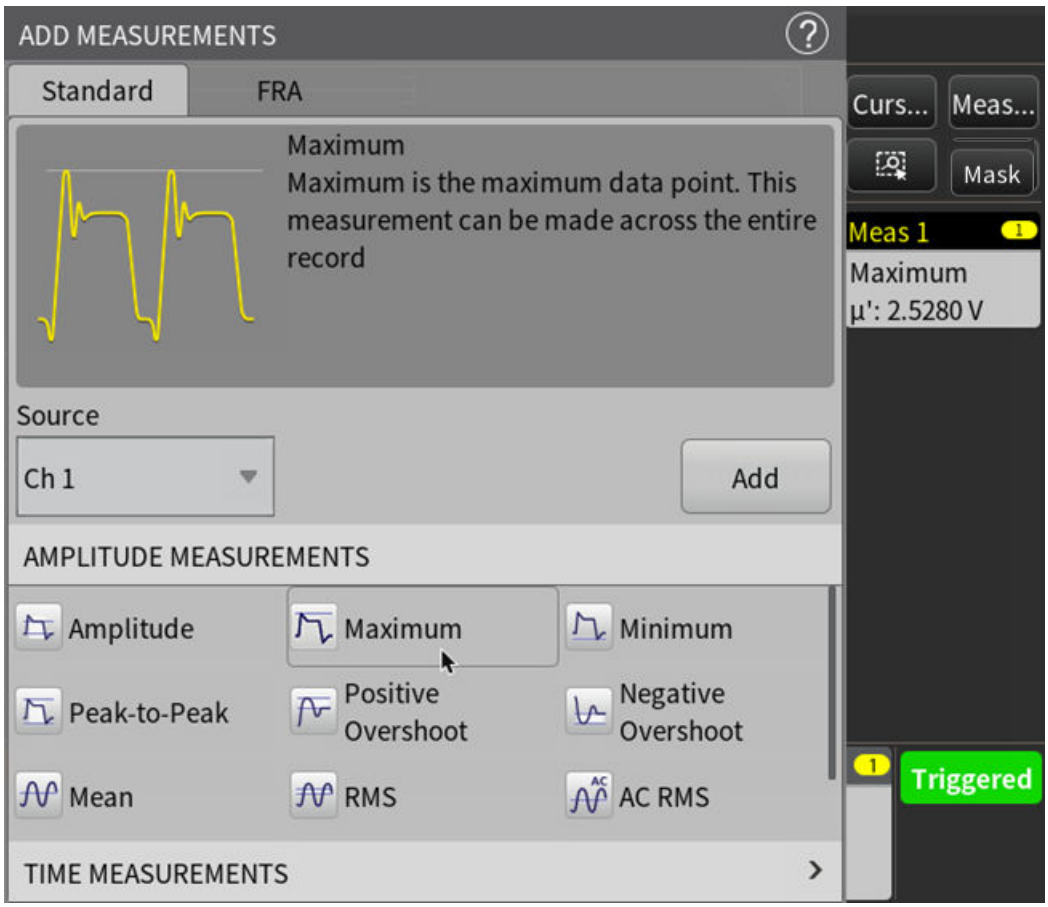
2. Tap the **Measure** button to open the **Add Measurements** configuration menu .

NOTE: If the menu shows tabs other than **Standard**, then optional measurement types have been installed on the instrument. Select a tab to show the measurements for that option.

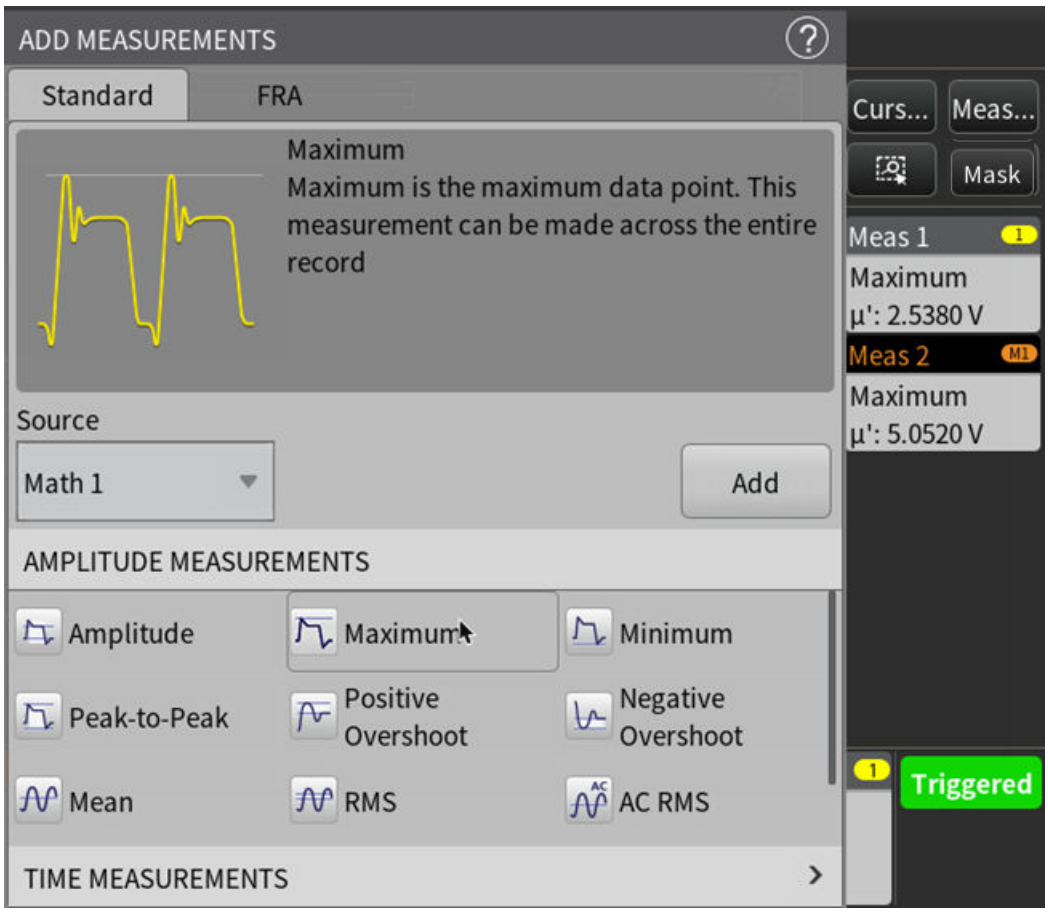
3. Tap the **Source** field and select the measurement source.



4. Select a measurement category panel, such as **Amplitude Measurements** or **Time Measurements**, to display measurements for those categories.
5. Select a measurement and tap **Add** to add the measurement to the **Results** bar. You can also double-tap a measurement to add it to the **Results** bar.



6. Select and add other measurements for the current source. Tap the measurement category panels to display and select other measurements to add.
7. To add measurements for other sources, select a different source, select a measurement, and add the measurement.

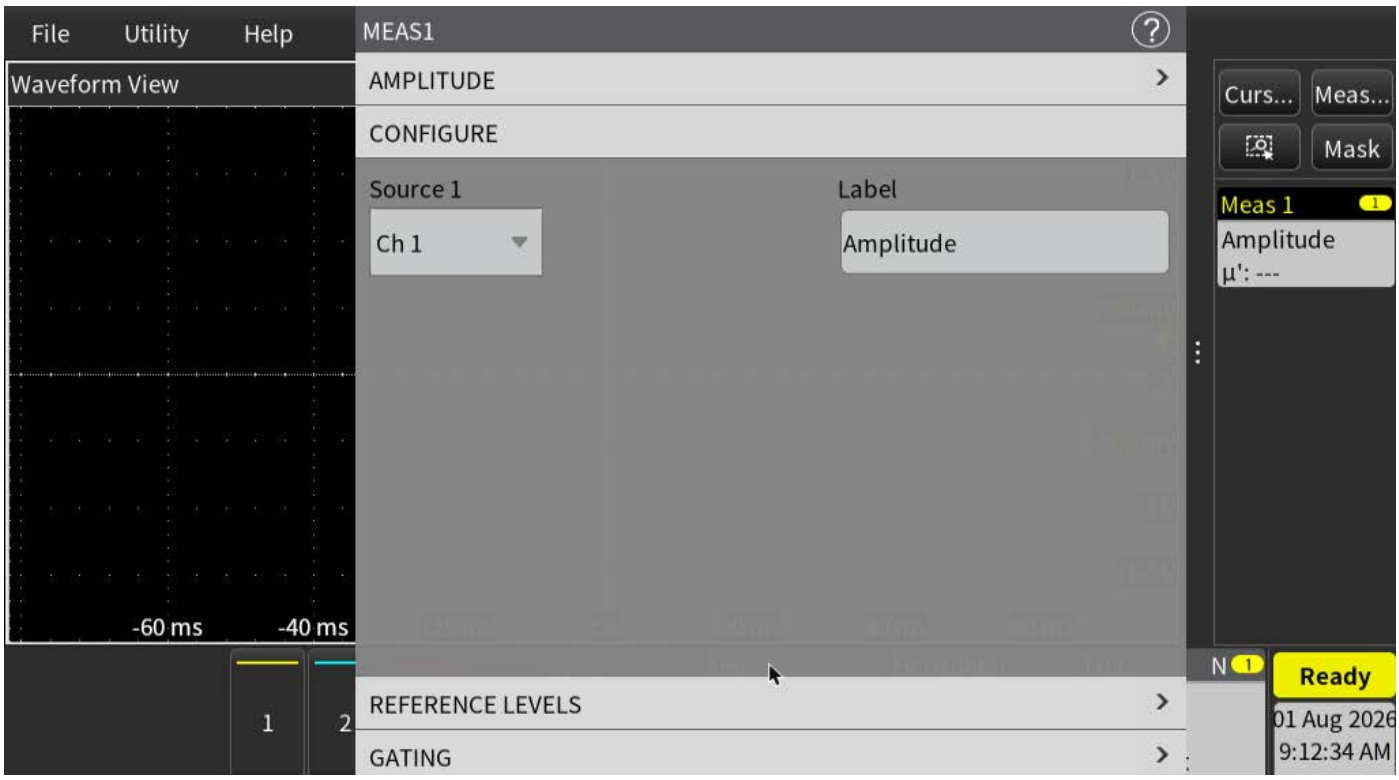


8. Tap outside the menu to close the menu.
9. To further adjust a measurement's settings, double-tap a measurement badge to open a configuration menu for that measurement. See Configure a measurement (on page 45).
10. Tap the Help icon on the menu title for more information on settings.

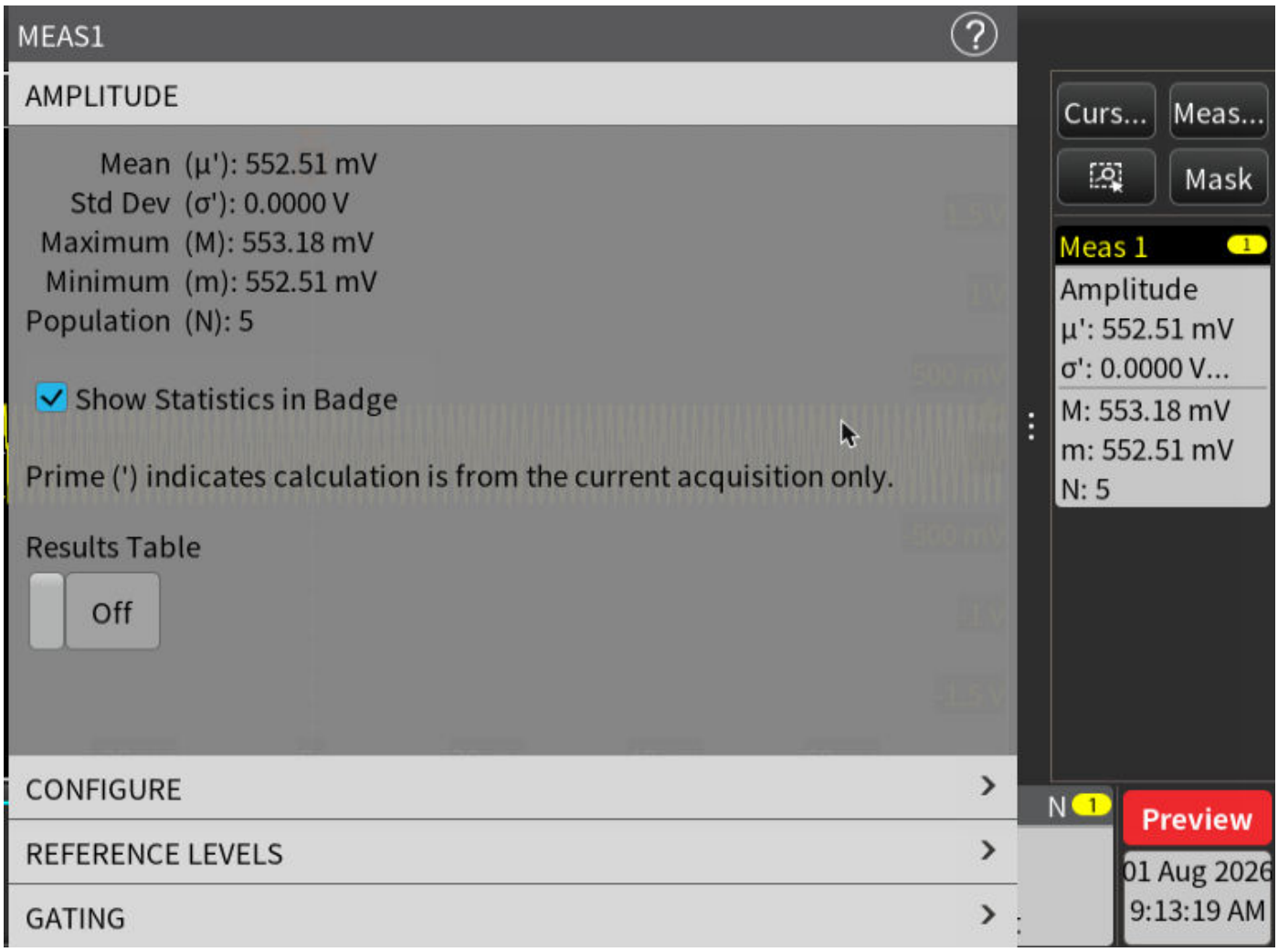
Configure a measurement

Use this procedure to add statistical readouts to the measurement badge, display plots for the measurement, and refine measurement parameters (configuration, global versus local scope of settings, gating, filtering, and so on).

1. Double-tap a measurement badge to open its **Measurement** configuration menu.



2. Tap **Show Statistics in Badge** to add statistical readouts to the measurement badge.

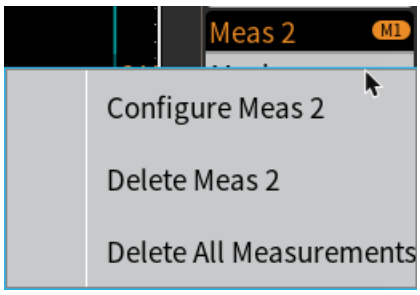


3. Tap available panel titles to make changes for those categories.
4. Use the available fields to refine the measurement conditions. Displayed fields depend on the measurement. Selection changes take effect immediately. Selection changes can also change fields in other panels.
5. Tap the Help icon on the menu title for more information on this menu's settings.
6. Tap outside the menu to close the menu.

Delete a Measurement badge

Use this procedure to remove a Measurement badge from the Results bar.

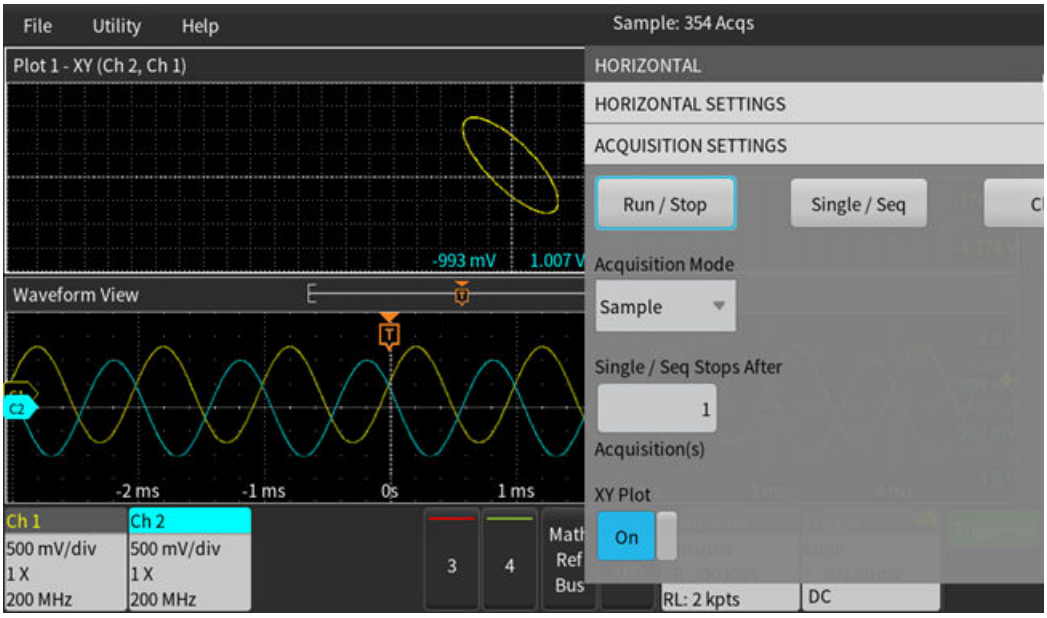
1. Touch and hold the Measurement badge that you want to delete. The instrument opens a right-click menu.
2. Select **Delete Meas** to delete that badge from the Results bar.



3. The second way to delete a Measurement badge is by dragging it off the right edge of the display.

Display an XY plot

Use this procedure to display an XY plot.



1. Double-tap **Horizontal** badge on the Setting bar.
2. Tap the **Acquisition Settings** panel.
3. Tap **XY** to **On**. The plot is added to the screen.
4. Double-tap in the Plot view to open a configuration menu for that plot.

Display an FFT math waveform

Use this procedure to display an FFT math waveform.

The FFT process mathematically converts the standard time-domain signal (repetitive or single-shot acquisition) into its frequency components. The FFT function processes the waveform record and displays the FFT frequency domain record, which contains the input signal frequency components from DC (0 Hz) to ½ the sample rate (also called the Nyquist frequency).

1. Tap **Math Ref Bus** and select one of the available math waveforms to create a math waveform and open the Math configuration menu.
2. Tap **Source** and select the signal source from the list.
3. Set Math Type to **FFT**. The FFT of the waveform is displayed in an FFT Math waveform view.
4. Double-tap on the FFT waveform display to open a configuration menu to further refine the FFT display.

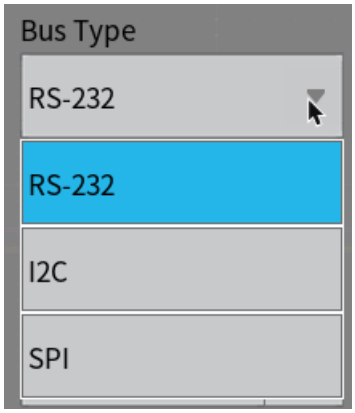
Add a serial bus to the Waveform view

Use this procedure to add a serial bus to the Waveform view.

This instrument supports decoding parallel buses (standard with the instrument) and several serial buses.

Use a Bus configuration menu to define a bus from which to acquire, decode, and display data.

1. Tap the **Math Ref Bus** > **Bus** button on the Settings bar to add a Bus badge to the Settings bar, add a bus waveform to the screen.
2. Double tap the **Bus** badge to open the Bus configuration menu
3. Tap **Bus Type** and select the bus type from the drop-down list.



4. Use the fields and controls to select the bus signal sources, thresholds, other parameters, and the output format. The decoded bus is updated on the screen as you make changes to the settings.

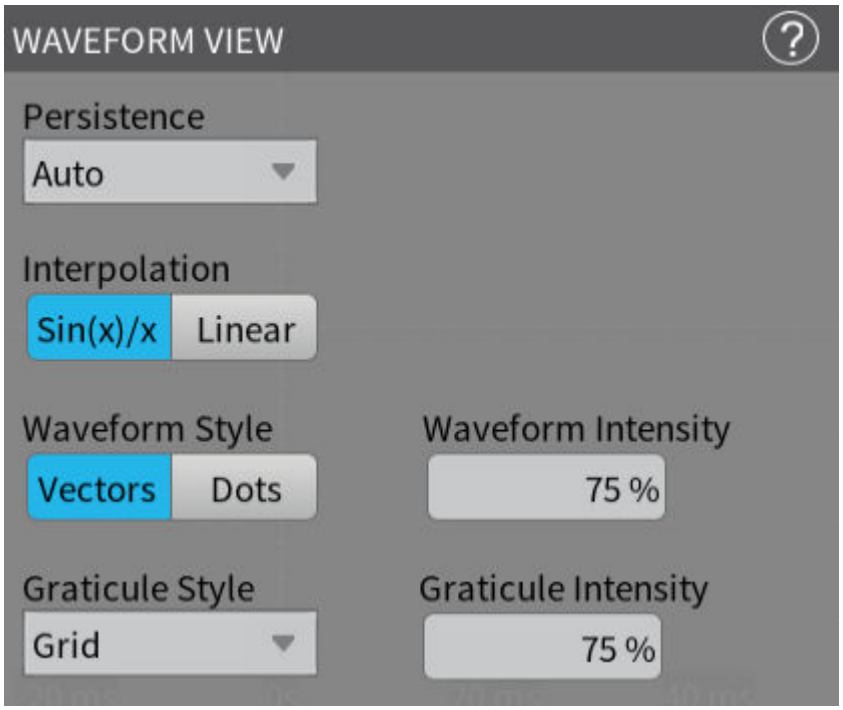


5. Tap outside of the Bus configuration menu to close it.
6. Double-tap the **Trigger** badge and use the Trigger configuration menu to trigger on a specific condition in the bus.
7. For more information on serial bus settings, see Bus configuration (on page 83).

Change waveform view settings

Use this procedure to change the waveform trace interpolation algorithm, waveform persistence, style and intensity, and graticule style and intensity.

1. Double-tap on an open graticule area to open the **Waveform View** configuration menu.

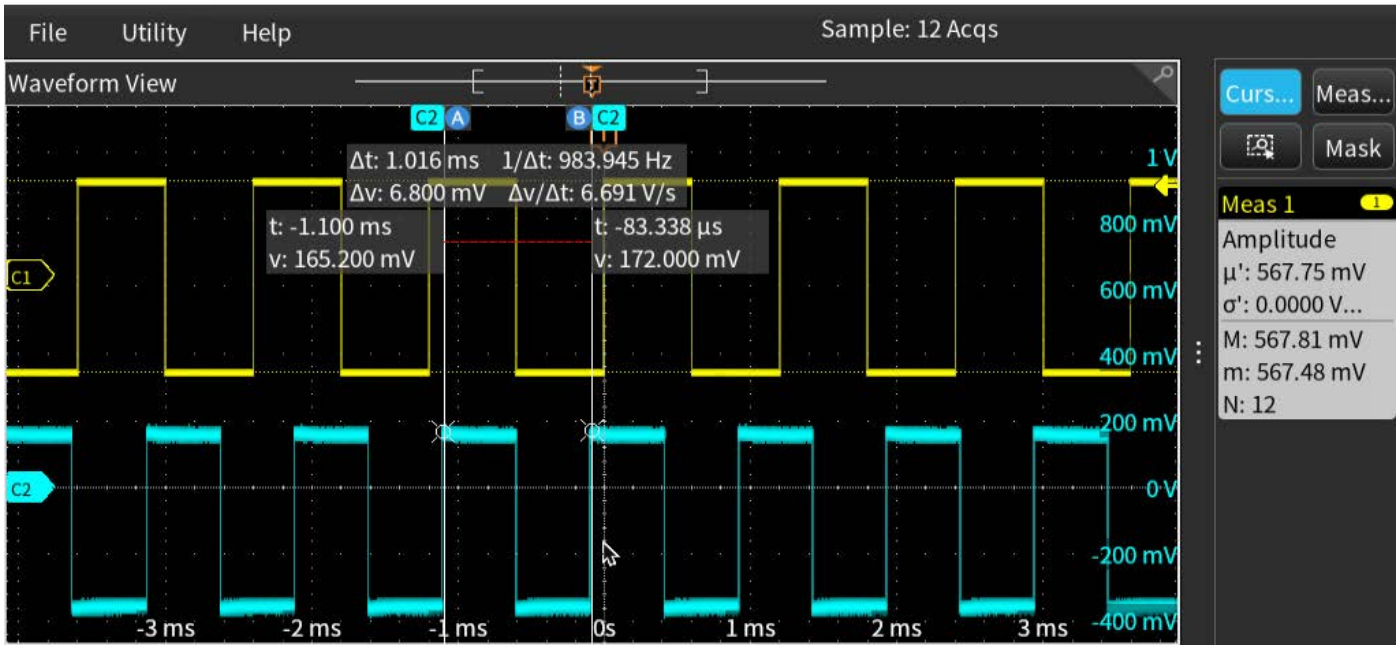


2. Use the controls to set the waveform interpolation algorithm, waveform point persistence, style and intensity, and graticule style and intensity.
3. See the Waveform View configuration menu (on page 116) Help topic for more information on the waveform view parameters.
4. Tap outside the menu to close the menu.

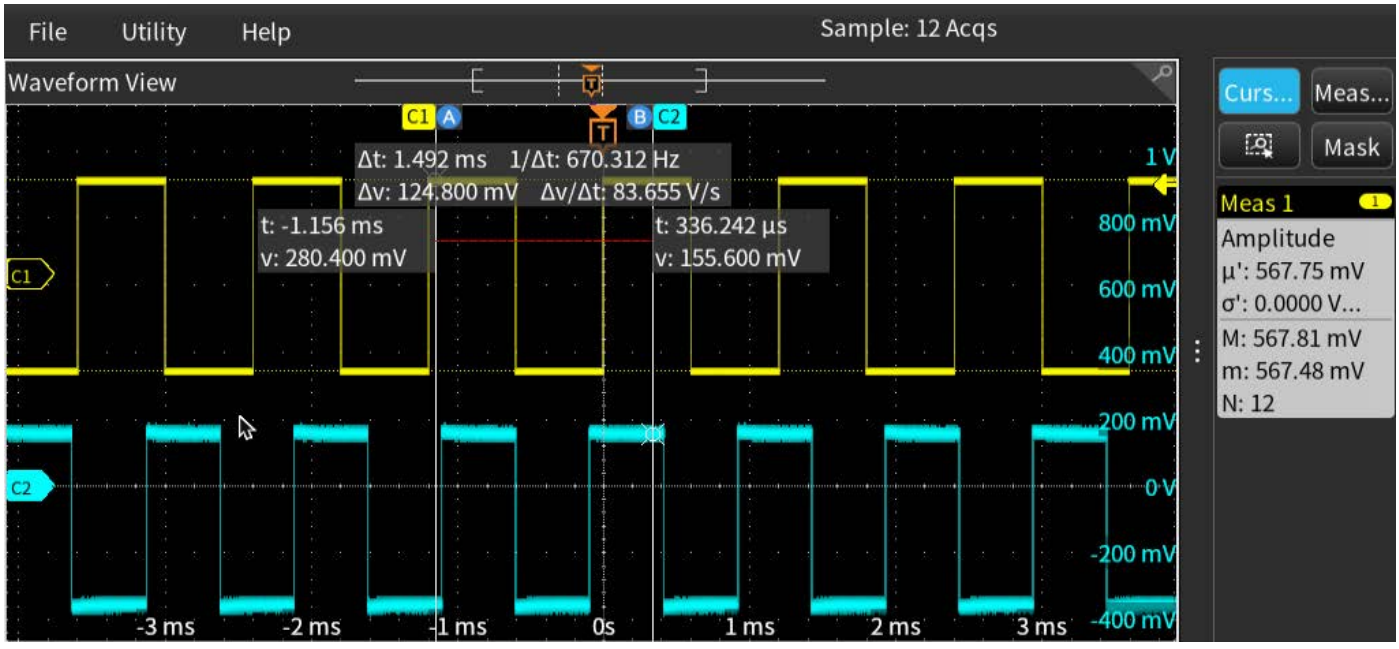
Display and configure cursors

Cursors are on-screen lines that you can move to take measurements on specific parts of a waveform or between two different waveforms. Cursor readouts show both current position values and the difference (delta) between cursors.

1. Tap the waveform or the channel to which you want to add cursors.
2. Tap the **Cursors** button. The cursors are added to the display.



3. Use Multipurpose Knobs **A** and **B** to move the cursors, or touch and drag a cursor. Cursors show readouts that show position and difference measurements between the cursors.
4. To move the cursors to a different channel or waveform, double-tap on the cursor label(s) and select a new source.
5. To further configure cursors, double-tap on either cursor line or the cursor readouts to open the **Cursors** configuration menu. For example, tap the Cursor type to select the cursors to display, such as Waveform, V Bars, H Bars, and V&H Bars. See Cursor configuration menu (on page 91).
6. To split the cursors between two waveforms, tap the **Source** field and select **Split** and select the source for each cursor. The cursors are moved to the specified waveforms.



7. To stop showing cursors, open the Cursors configuration menu and set Display to **Off** or tap the Cursors button in the upper right.

Using Default Setup

Use Default Setup to restore instrument settings to their factory defaults.

1. Press the front panel **Default Setup** button to return the instrument to its factory default settings (horizontal, vertical, scale, position, and so on).
2. You can also select **File** > **Default Setup** to restore default settings.

Advanced triggering

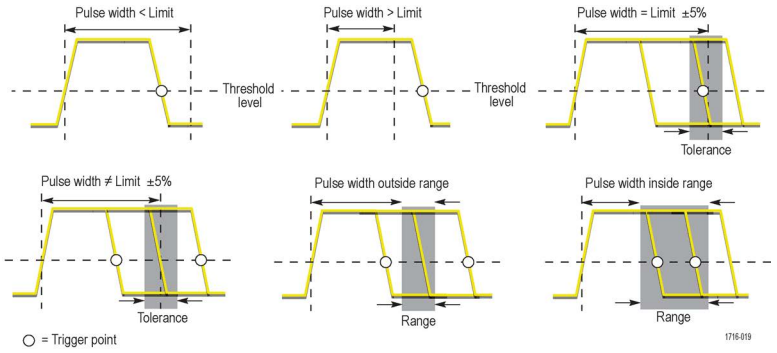
You can check the advanced trigger status in the trigger menu. The menu indicates the trigger type and then shows sources, levels, or any other parameters that are important for the particular trigger type.

Trigger on a pulse width event

Pulse-width triggering triggers the oscilloscope when a signal pulse width is less than, greater than, equal to, or not equal to a specified pulse width. This trigger is useful for digital logic troubleshooting.

To trigger on a pulse width event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Pulse Width**. See Pulse Width Trigger configuration menu (on page 109).
3. Tap **Source** and select the trigger source.
4. Tap **Trigger When** and select the pulse width condition on which to trigger (> Limit, < Limit, Inside Range).



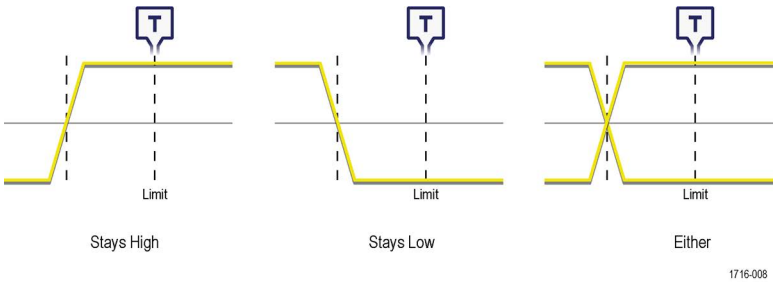
5. Set the pulse width time constraints:
   a. For >Limit and < Limit, tap the **Time Limit** field and use the assigned multipurpose knob to set the pulse width time condition to meet.
   b. For Inside Range, tap the **High Time Limit** and **High Time Limit** fields and use the assigned multipurpose knobs to set the pulse width time range condition to meet.
6. Tap the Level field and set the threshold value at which you want to measure pulse width.
7. Select the pulse polarity on which to trigger.

Trigger on a timeout event

Use Timeout triggering to trigger the oscilloscope when a signal stays above, stays below, or stays above or below a specified level for longer than a specified time limit.

To trigger on a timeout event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Timeout**. See Timeout Trigger configuration menu (on page 110).
3. Tap **Source** and select the signal source.
4. Tap **Trigger When** and select the condition on which to trigger (**Stays High**, **Stays Low**, or **Either**).



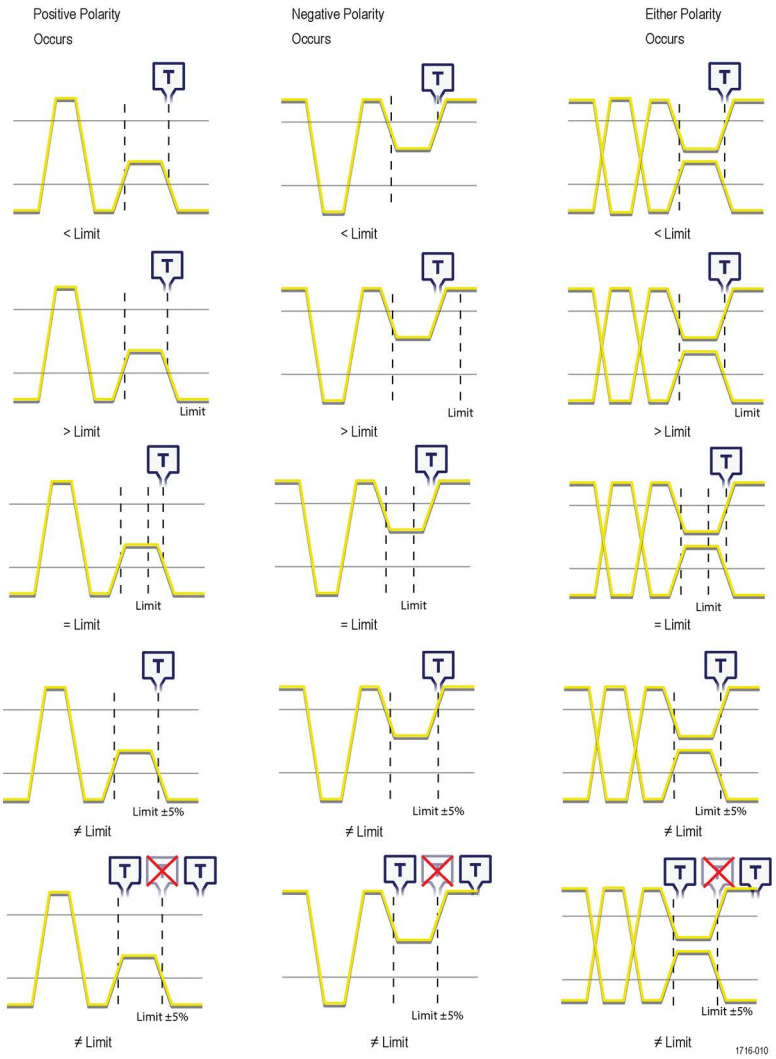
5. Tap the **Threshold** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the threshold value that determines when the signal is high or low.
6. Tap the **Time Limit** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the maximum time that a signal can stay high or low before transitioning to the opposite state.

Trigger on a runt signal event

Use Runt triggering to locate signal pulses that do not make a full logic level transition.

To trigger on a runt event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Runt**. See Runt Trigger configuration menu.
3. Tap **Source** and select the trigger source.
4. Tap **Trigger When** and select the condition on which to trigger. Available options are >Limit, <Limit, and Occurs.



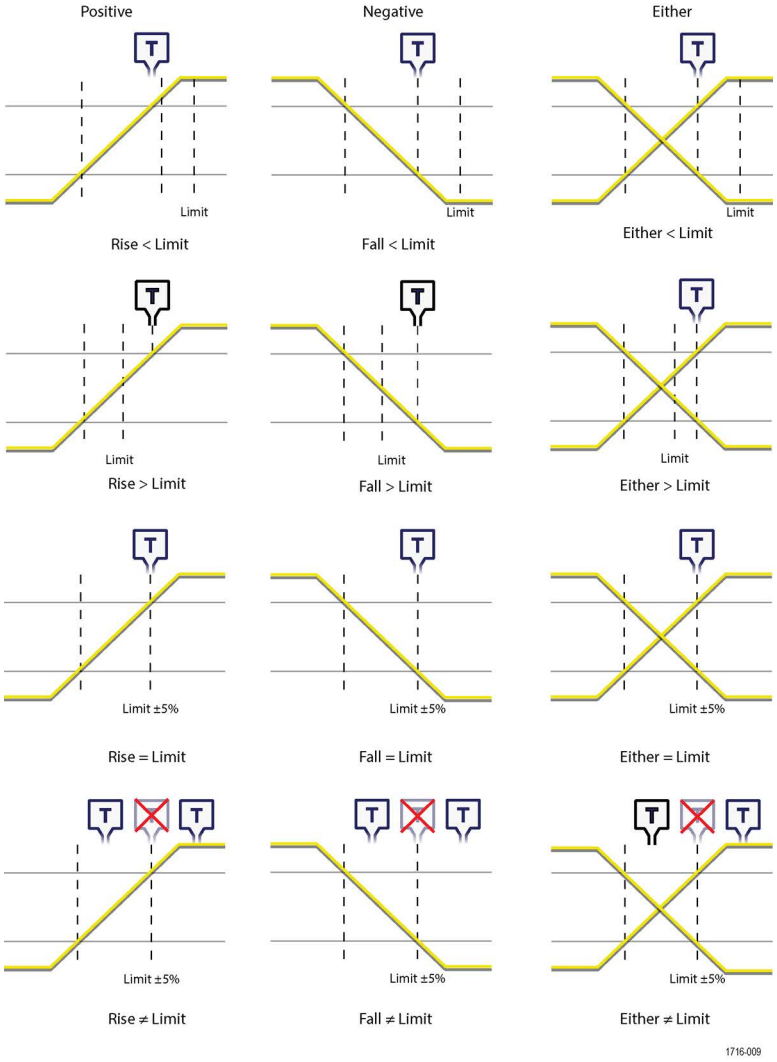
5. Tap the **Upper Threshold** field and use the assigned multipurpose knob to set the threshold value that defines the upper limit that the signal must cross to be a valid signal transition (or double-tap the field and use the keypad to enter a value).
6. Tap the **Lower Threshold** field and use the assigned multipurpose knob to set the threshold value that defines the lower limit that the signal must cross to be a valid signal transition (or double-tap the field and use the keypad to enter a value).
7. If **Trigger When** is set to a limit value, use the **Time Limit** field to set the runt signal time constraints.

Trigger on a rise or fall time event

Use the Rise/Fall Time trigger to trigger when the rise or fall time of a signal is less than, greater than, or In Range.

To trigger on a Rise/Fall event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Rise/Fall**. See Rise/Fall Time Trigger configuration menu (on page 112).
3. Tap **Source** and select the trigger signal source.
4. Tap **Trigger When** and select the condition on which to trigger.



5. Tap the **Time Limit** field to set the rise or fall time constraints.
6. Tap the **Upper Threshold** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the length of setup time that the data signal must be stable before a clock edge occurs.
7. Tap the **Lower Threshold** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the length of hold time that the data signal must be stable after a clock edge occurs.
8. Use the **Slope** controls to set the signal transition direction to detect. (rising or falling).

Trigger on a serial bus

Use this procedure to set up triggering on a serial bus.

Use this procedure if you have already created a serial bus. To trigger on a Rise/Fall event:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Tap **Trigger Type** and select the bus type from the list.
3. Tap **When** and select what to trigger on from the list. The displayed fields and controls depend on the bus type and **When** selection. Use these fields to trigger on a specific bus condition. See Bus Trigger configuration (on page 112).

Setting waveform display parameters

Use waveform display controls to set the display mode, persistence, style, and intensity display parameters, and graticule style and intensity.

Set the Waveform Interpolation mode

Use this procedure to set the waveform interpolation mode, which sets how waveform data points are calculated between record data points. This selection is used for both screen display interpolation and built-in math function interpolation.

1. Double-tap on an open graticule area to open the Waveform View configuration menu.
2. Tap the buttons under **Interpolation** to select either Sin(x)/x or Linear.

   Sin(x)/x calculates record points along a curve between the actual acquired samples. This form of interpolation is useful when acquiring rounded waveforms such as sine waves. It is good for general-purpose uses but may introduce overshoot or undershoot in signals with fast rise times. This interpolation is also useful for looking at high-frequency signals, especially where the frequency components are just below the Nyquist frequency.

   Linear calculates record points between actual acquired samples using a straight-line fit. This interpolation is useful for measuring waveforms with fast rise times, such as pulse trains.

Set the waveform persistence, style, and intensity

Use the Waveform View configuration menu to set waveform persistence, style, and intensity.

1. Double-tap on an open graticule area to open the Waveform View menu.
2. Tap the **Persistence** field to select the persistence option.
   a. **Off** disables display persistence.
   b. **Auto** lets the oscilloscope automatically determine a persistence time for you.
   c. **Infinite** persistence continuously accumulates record points until you change one of the acquisition display settings. Use infinite persistence for displaying unique signal anomalies, such as glitches.
   d. **Variable** persistence accumulates record points for a specified time interval. Each record point decays independently according to the time interval. Use variable persistence for displaying infrequently appearing signal anomalies, such as glitches.

      If you select Variable persistence, tap **Variable Persistence Time** and set the time using the multipurpose knob, or double-tap the field and use the virtual keypad to enter the time value.
3. Tap the Waveform Style buttons to set waveforms to draw as vectors (continuous lines) or dots.
   - **Vectors** displays the waveform with the waveform sample values connected using the selected interpolation method.
   - **Dots** displays the individual waveform sample values with no interpolation.
4. Tap the **Waveform Intensity** field and use the multipurpose knob to set the brightness of all waveforms.

Set the graticule style and intensity

Use this procedure to set the graticule (display grid) style and intensity.

1. Double-tap on an open graticule area to open the Waveform View configuration menu.
2. Tap the **Graticule Style** field to select a graticule style from the list.

- **Grid** provides a grid, cross hairs, and frame on the instrument display.
- **Full** shows a frame and a grid on the instrument display. This style is useful for making quick, full-screen measurements with cursors and automatic readouts when cross hairs are not needed.
- **None** provides a frame without a grid and cross hairs.

3. Tap the Graticule Intensity field and use the multipurpose knob to set the brightness of all graticules.

Mask testing

Mask testing determines how well your signals conform to expected signal quality and provides instant automated statistical analysis. A mask defines a portion of the instrument display that a signal must not enter. Mask testing determines how well your signals conform to expected signal quality and provides instant automated statistical analysis. A mask defines a portion of the instrument display that a signal must not enter. Mask testing is supported forvertical and horizontal tolerance.

Create a Mask on waveforms

Use the **Mask** button to create a new mask on the instrument display.

1. Tap the **Mask** button. This will create a waveform tolerance mask using the current acquisition as the desired waveform.
2. Double-tap the mask area on the display to bring up the **Mask Definition** menu. See the Mask Badge configuration menu.

Zoom in on waveforms

Use the zoom tools to magnify waveforms to view signal details.

Turn on Zoom mode

Zoom mode lets you look at a portion of your waveform in greater detail. Enable Zoom mode and touch and drag on-screen to select the area to zoom.

1. To enable Zoom mode, use one of the following methods:
2. Tap the **Zoom** button.
3. Touch and drag in the Waveform view to draw a box on the area of interest.
4. Tap the Zoom icon in the corner of Waveform and Plot views.



5. Zoom overview:
   a. To use Zoom once enabled, tap the zoom icon touch and draw a box around an area of interest in the Waveform or Plot view to immediately display the zoomed waveform and the Zoom Overview window.
   b. You can draw boxes in the Waveform view, most plots, and the Zoom Overview area.
   c. To exit the zoom display mode, tap the Zoom icon in the corner of the Waveform view and plot.

For more information on zoom, see the Zoom user interface (on page 27) topic.

Measurement configuration

After adding a measurement, you can customize the measurement for more precise results by using gating, setting reference levels, or adding a label.

To customize measurements, double-tap a Measurement badge in the Results bar to open the Measurement configuration menu (on page 74).

Customize the following elements:

- **Label** your measurement to clarify documentation.
- **Reference levels** determine how time-related measurements are taken.
- **Gating** confines the measurement to a certain portion of a waveform.

Label a measurement

Use this procedure to add a custom label to a measurement.

Measurement labels appear on the Measurement badge and can be added to Results tables.

Labels are set in the Measurement panel of the Measurements configuration menu. See Measurement configuration menu (on page 74) overview.

Prerequisite: To set measurement reference levels you must be taking a measurement. See Add a measurement.

1. Double-tap a **Measurement** badge in the Results bar to open the Measurement configuration menu (on page 74).
2. Tap the **Configure** panel.
3. Double-tap the **Label** field and enter the label text using the virtual keyboard, then tap the **Enter** button to add the label text to the Measurement badge. Tap outside the virtual keyboard to close it.

   NOTE: The new label text replaces the default measurement name on the Measurement badge that shows the measurement type (Frequency, Peak-to-Peak, and so on). To view the measurement type for a relabeled measurement, double-tap the Measurement badge and look at the topmost panel name, which has the measurement name.

   Custom measurement labels can be added to a Results table.
4. Tap anywhere outside the Measurement menu to close it.

Set measurement reference levels

Use this procedure to set measurement reference levels.

Reference levels are set in the Reference Levels panel of the Measurements configuration menu. See Measurement configuration menu (on page 74) overview.

Prerequisite: To set measurement reference levels you must be taking a measurement. See Add a measurement (on page 42).

Procedure

1. Double-tap a Measurement badge. The Measurement configuration menu is displayed.
2. Tap the **Reference Levels** panel.
3. Tap **Set Levels In** and select either **%** or **Absolute**.
   - **%** sets the High, Mid, and Low reference levels as percentages of the calculated Top and Base signal levels. Tap the Levels **10% - 90%**, **20% - 80%**, or **Custom** button to select the type of percent value to set.
     - **10% - 90%** sets the Low, Mid and High Ref values to 10%, 50% and 90% respectively for both rising and falling edges.
     - **20% - 80%** sets the Low, Mid and High Ref values to 20%, 50% and 80% respectively for both rising and falling edges.
     - **Custom** opens the Rising and Falling Edge numeric entry fields to set High, Mid and Low Ref levels to different values for the rising and falling edges of a waveform. Tap the Rising Edge and Falling Edge **High**, **Mid**, or **Low** fields and set the level using the assigned multipurpose knob. Or double-tap the field and use the virtual keypad to enter a value.
   - **Absolute** sets the High, Mid, and Low reference levels to specific signal levels. Tap the Levels **Same** or **Unique** buttons to select the type of absolute value to set.
     - **Same** sets the High, Mid, and Low reference for both rising and falling edges of a signal to the specified values. Tap the Threshold **High**, **Mid**, or **Low** field and set the threshold using a multipurpose knob. Or double-tap the field and use the virtual keypad to enter a value.
     - **Unique** opens the Rising and Falling Edge numeric entry fields to set High, Mid and Low Ref levels to different values for the rising and falling edges of a waveform. Tap the Rising Edge and Falling Edge **High**, **Mid**, or **Low** fields and set the level using the assigned multipurpose knob. Or double-tap the field and use the virtual keypad to enter a value.
4. Tap the **Base Top Method** drop down tab and select the method from the list. This setting is only available when using the % level setting.
5. Touch **Hysteresis** and use the assigned multipurpose knob to set the value.
6. Tap anywhere outside the Measurement configuration menu to close it.

Set measurement gates

Use this procedure to specify which portion of your waveform is used to take measurements.

Gating is set in the **Gating** panel of the **Measurements** configuration menu. See Measurement configuration menu (on page 74) overview.

To set measurement gates you must be taking a measurement. See Add a measurement (on page 42)t.

Procedure

1. Double-tap a measurement badge to open the Measurement configuration menu.
2. Tap the **Gating** panel.
3. Tap the **Gating Type** drop down tab and select the gating type from the list:
   - **None** takes measurement across the entire waveform record.
   - **Screen** takes measurements on that portion of the waveform shown in the display. When Zoom is on, the display is the zoom window.
   - **Cursors** takes measurements on that portion of the waveform between the cursors. Selecting Cursors opens cursors on the measurement source. Set the cursors so that the waveform area of interest is between the cursors.
4. Tap anywhere outside the Measurement configuration menu to close it.
5. To return measurement gating to its default value, open the **Gating** panel and select a gating type of **Screen**.

NOTE: Measurement gating is a global setting and will affect all measurements. Changing the gating in one measurement will change it in all measurements.

Saving and recalling information

Use these procedures to save or recall waveforms, setups, or sessions.

Save a screen image

Use this procedure to save a screen image.

1. Tap the **File** menu and select **Save As**.

   The Save As configuration menu opens.

2. Tap **Screen Capture** to open the Screen Capture tab.
3. Tap **Save Location** or **Browse** to select the location to save the file.
   a. Tap the drop down arrow in the **Save Location** field and select the location to save the file from a list of recent save locations. Or double-tap the field and use the virtual keyboard to enter a path to the save location.
   b. Tap **Browse** to open the Browse Save As Location configuration menu to navigate to and select a location at which to save the file.
4. Tap **File Name** shows the name last used to save a file. The default name is Tek000. To change the file name, double-tap the file name and enter a new file name using the virtual keyboard.
5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.
   - If Auto Increment File Name is enabled the **Count** defaults to 000 if there are no files at the specified location and file name that already use incremented file names. If there are files at the save location that already use the specified file name, and have already been saved using count increments, the Count field shows the next count value that will be added to the file name when the file is saved.
   - To change the starting count value, tap the Count field and use the assigned knob to change the value, or double-tap the field and use the virtual keypad to change the value.
6. Tap **Save As Type** and select the desired graphic image file type from the list.
7. Tap **Save** to save the screen image to the specified file name, location, and type.

NOTE: Once you have saved a file using the Save As configuration menu, you can push the front-panel **Save** button to immediately save the same type file again, without opening any menus.

Save a waveform or spectrum trace to a file

Use this procedure to save channel waveform data to a comma-separated values (csv) or Tektronix waveform data (wfm) file, for later analysis or inclusion in reports. You can also save to files.

1. Tap the **File** menu and select **Save As** and the Save As configuration menu opens.
2. Tap **Waveform** to open the Waveform tab.
3. Tap **Save Location** or **Browse** to select the location to save the file.
   a. Tap the **Save Location** drop down arrow and select the location to save the file from a list of recent save locations. Or double-tap the field and use the virtual keyboard to enter a path to the save location.
   b. Tap **Browse** to open the Browse Save As Location configuration menu to navigate to and select a location at which to save the file.
4. **File Name** shows the name last used to save a file. The default name is Tek000. To change the file name, double-tap the file name and enter a new file name using the virtual keyboard.
5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.
   - If Auto Increment File Name is enabled the **Count** defaults to 000 if there are no files at the specified location and file name that already use incremented file names. If there are files at the save location that already use the specified file name, and have already been saved using count increments, the Count field shows the next count value that will be added to the file name when the file is saved.
   - To change the starting count value, tap the Count field and use the assigned knob to change the value, or double-tap the field and use the virtual keypad to change the value.
6. Tap **Save As Type** and select the desired waveform data type from the list.
7. Tap **Source** and select the source of the waveform to save from the list.
8. Tap **Save** to save the waveform to the specified file name, location, and type.

NOTE: Once you have saved a file using the **Save As** configuration menu, you can push the front-panel **Save** button to immediately save the same type file again, without opening any menus.

Save instrument settings to a file

Use this procedure to save instrument settings to a Tektronix setup (.set) file.

1. Tap the **File** menu and select **Save As**.

   The Save As configuration menu opens.
2. Tap **Setup** to open the Setup tab.
3. Tap **Save Location** or **Browse** to select the location to save the file.
   a. Tap the **Save Location** drop down arrow and select the location to save the file from a list of recent save locations. Or double-tap the field and use the virtual keyboard to enter a path to the save location.
   b. Tap **Browse** to open the Save As browse location to navigate to and select a location at which to save the file.
4. **File Name** shows the name last used to save a file. The default name is Tek000. To change the file name, double-tap the file name and enter a new file name using the virtual keyboard.
5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.
   - If Auto Increment File Name is enabled the **Count** defaults to 000 if there are no files at the specified location and file name that already use incremented file names. If there are files at the save location that already use the specified file name, and have already been saved using count increments, the Count field shows the next count value that will be added to the file name when the file is saved.
   - To change the starting count value, tap the Count field and use the assigned knob to change the value, or double-tap the field and use the virtual keypad to change the value.
6. Tap **Save** to save the setup information to the specified file name and location.

NOTE: Once you have saved a file using the Save As configuration menu, you can push the front-panel **Save** button to immediately save the same type file again, without opening any menus.

You can you can push the front-panel **Save** button to immediately save the same type file again, without opening any menus

Recall a reference waveform

Use this procedure to recall (load) and display a saved waveform as a Reference waveform. Up to two reference waveforms can be loaded and displayed..

1. Tap the **Math Ref Bus** and select either **Ref 1** or **Ref 2** on the Settings bar.

   The Recall configuration menu opens to the last location used to recall a waveform file. If the location is not available, the menu defaults to the instrument C drive.

2. Tap **Files of Type:** and select the file type from the list.

   The files list updates to show all files that match the selected file type.

3. Tap the file name in the list to add the file to the **File Name** field.

4. Tap **Open** to exit the menu and add the reference waveform to the screen.

5. If the file location shown in the **Look In** field does not contain the file to recall, navigate to the folder that contains the file to recall, using one of the following methods:
   - To recall files from recently accessed file locations, tap the drop down arrow in the **Look in** field and select from the list of recently accessed locations.
   - Tap in the **Drive** column and use the displayed contents to navigate to locations on the internal storage location C or a connected USB drive.
   - Double-tap the **Look in** field and use the virtual keyboard to manually enter a path to the file location (folder).

   As you navigate the folders, the files list area shows all files that match the file type selected in the Files of Type field.

6. Select the file to recall, using one of the following methods:
   - If the file was recently recalled, tap the drop down arrow in the **File Name:** field and select from a drop-down list of recently recalled files.
   - Select a file name in the main files list. The file **Name** field updates to show the selected file name.

7. Tap **Recall**.

   The reference waveform is loaded and displayed, and a **Ref** badge is added to the Settings bar.

Recall a Setup file

Use this procedure to recall (load) and configure instrument settings from a Setup file.

1. Select **File > Recall** from the Menu bar to open the Recall configuration menu.

2. Tap **Setup** to open the Setup tab.

   The Recall configuration menu opens to show the settings last used to recall a setup file.

3. Tap **Files of Type:** and select the file type from the list.

   The files list updates to show any files that match the selected file type.

4. Navigate to the folder that contains the file to recall, using one of the following methods:
   - To recall files from recently accessed file locations, tap the drop down arrow in the **Look in** field and select from the list of recently accessed locations.
   - Tap in the **Drive** column and use the displayed contents to navigate to locations on the internal storage location C, or on a connected USB drive.
   - Double-tap the **Look in** field and use the virtual keyboard to manually enter a path to the file location (folder).

As you navigate the folders, the files list area shows all files that match the file type selected in the Files of Type field.

5. Select the file to recall, using one of the following methods:
   - If the file was recently recalled, tap the drop down arrow in the **File Name:** field and select from a drop-down list of recently recalled files.
   - Select a file name in the main files list. The File Name field updates to show the selected file name.
6. Tap **Recall**.

   The instrument loads the setup file and reconfigures the oscilloscope to the setup file settings.

NOTE: Only setting files saved on the 1 Series oscilloscope can be recalled.

Menus and dialog boxes

The topics in this section provide information about all of the menus on your instrument and the available settings within each menu.

Add Measurements configuration menu overview

Use this configuration menu to select measurements you want to take on waveforms and add the measurements to the Results bar.

To open the **Add Measurements** configuration menu, tap the **Measure** button in the **Analysis** controls area.

To add a measurement, select the measurement type tab, select the input source or sources, select the measurement, and either tap the **Add** button or double-tap the measurement. The measurement is added to the Results bar.

To change individual measurement settings, double-tap the Measurement badge to open a configuration menu for that measurement. See Measurement configuration menu (on page 74) overview.

Add Measurements menu fields and controls

Field or control	Description
Measurement panels	The tabs in various measurement panels organize measurements by their type. The available tabs are **Standard** and **FRA**.
Measurement description (graphic and text)	Shows a graphic and short description of the selected measurement. Use this information to verify that the selected measurement is correct for what you want to measure.
Source	Selects the measurement source.
Add button	Adds the selected measurement as a measurement badge to the Results bar. You can also double-tap a listed measurement to add it to the Results bar.

Measurement panels

Panel	Description
Amplitude Measurements panel	This panel contains various amplitude measurement tabs. Tap a tab and touch and drag the list to scroll through all measurements. See Amplitude Measurements panel (on page 68).
Time Measurements panel	This panel contains various time measurement tabs. Tap a tab and touch and drag the list to scroll through all measurements. See Time Measurements panel (on page 69).

Standard measurements tab

The **Standard** tab lists the standard set of amplitude and time measurements.

To open the **Standard** tab, tap the **Measure** button and tap the **Standard** tab.

To add a measurement to the Results bar:

1. Select the signal source.
2. Tap a measurement panel:
   - Amplitude Measurements panel (on page 68)
   - Time Measurements panel (on page 69)
3. Select a measurement. If a measurement requires two signal sources, select the sources in the Source 1 and Source 2 fields.
4. Tap **Add**. You can also double-tap a measurement to add it immediately to the Results bar.

Amplitude Measurements panel

The **Amplitude Measurements** panel lists available amplitude-related measurements that you can take on channel signals, math waveforms (time-domain), and reference waveforms. The measurement annotations are supported for all amplitude measurements.

To open the Amplitude Measurements panel:

1. Tap the **Measure** button.
2. Tap the **Amplitude Measurements** panel.

To add a measurement to the Results bar:

1. Select the signal **Source**.
2. Select a measurement.
3. Tap **Add**. You can also double-tap a measurement to add it immediately to the Results bar.

The Amplitude Measurements panel measurements

Measurement	Description
AC RMS	The true Root Mean Square voltage, minus any DC component, of the waveform data points that are above the Mean signal level.
Amplitude	The difference between the Top value and the Base value. **NOTE:** Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated.
Area	The area between the waveform and the Mean value, calculated by integrating the data points. The area above ground (0 V) is positive, while the area below ground is negative.
Base	The most common data point value below the midpoint of the waveform, over the measurement region. The algorithm to determine this value is selected with the Base Top Method. Base is used as the 0% value whenever low reference values are calculated, such as in rise time or fall time measurements. **NOTE:** Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated.
Maximum	The maximum data point value.

Measurement	Description
Mean	The arithmetic mean of all data points over the measurement region.
Minimum	The minimum data point value.
Negative Overshoot	The difference between the Minimum and Base values, divided by the Amplitude, and multiplied by 100 to express the measurement as a percentage of amplitude. **NOTE:** Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated.
Peak-To-Peak	The absolute difference between the Maximum and Minimum amplitudes in the measurement region.
Positive Overshoot	The difference between the Maximum value and the Top value, divided by the Amplitude, and multiplied by 100 to express the measurement as a percentage of amplitude. **NOTE:** Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated
RMS	The true Root Mean Square (The square root of the mean (average) value of the squared function of the waveform data points.)
Top	The most common data point value above the midpoint of the waveform over the measurement region. The algorithm to determine this value is selected with the Base Top Method. Base is used as the 100% reference value whenever high reference values are calculated, such as in rise time or fall time measurements. **NOTE:** Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated

See also:

Measurement configuration menu (on page 74) overview

Timing Time Measurements panel

Use the **Time Measurements** panel to add timing-related measurements to the Results bar. Time measurements can be taken on time-domain analog, math, and reference waveforms. The measurement annotations are supported for all time measurements.

To open the Time Measurements panel:

1. Tap the **Measure** button.
2. Tap the **Time Measurements** panel.

To add a measurement to the Results bar:

1. Select the signal source.
2. Select a measurement. If a measurement requires two signal sources, select the sources in the Source 1 and Source 2 fields.
3. Tap **Add**. You can also double-tap a measurement to add it immediately to the Results bar.

Time Measurements panel

Measurement	Description
Burst Width	The duration of a series of adjacent crossings of the Mid reference level. Bursts are separated by a specified idle time. The measurement is taken on each burst in a waveform record.
Fall Time	The time required for the trailing edge of the first pulse in the measurement region to fall from the high reference value (default = 90%) to the low reference value (default = 10%). The measurement is taken on each cycle of the waveform record.
Frequency	The frequency of the waveform. Frequency is the reciprocal of Period (Frequency = 1/Period).
Hold Time	The time between the specified Mid reference level crossing on the clock signal to the closest Mid reference level crossing on the specified data signal. The measurement is taken on each specified clock edge in the waveform record.
Negative Duty Cycle	The ratio of the negative pulse width to the signal period, expressed as a percentage. The measurement is taken on each cycle in the waveform record or measurement region.
Negative Pulse Width	The distance (time) between the mid reference (default 50%) amplitude points of a negative pulse. The measurement is taken on each cycle in the waveform record or measurement region.
Period	The time between two adjacent crossings of the Mid reference level (one cycle) of the waveform. The measurement is taken on each cycle of the waveform record or measurement region.
Positive Duty Cycle	The ratio of the positive pulse width to the signal period, expressed as a percentage. The measurement is taken on each cycle in the waveform record or measurement region.

Measurement	Description
Positive Pulse Width	The distance (time) between the mid reference (default 50%) amplitude points of a positive pulse. The measurement is taken on each cycle in the waveform record or measurement region.
Rise Time	The time required for the leading edge of the first pulse in the measurement region to rise from the low reference value (default = 10%) to the high reference value (default = 90%). The measurement is taken on each cycle of the waveform record.
Setup Time	The time between the specified Mid reference level crossing on the data signal to the closest Mid reference level crossing on the specified clock signal. The measurement is made on each specified clock edge in the waveform record.

See also

Measurement configuration menu (on page 74) overview

The PowerFRA tab

The FRA tab lists the optional Frequency Response Analysis (FRA)-related measurement that you can add to the Results bar.

NOTE: The FRA tab is only available on S model instruments.

To open the FRA measurement tab:

1. Tap the **Measure** button.
2. Tap the **FRA** tab.

To add a measurement to the Results bar:

1. Select the signal sources. If the measurement requires two signal sources, select the sources in the Source 1 and Source 2 fields.
2. Select the **Control Loop Response (Bode)** measurement.
3. Tap **Add**. You can also double tap a measurement to add it to the Results bar.

Control Loop Response (Bode) measurement configuration panel

Double-tap a Power measurement badge to open the Control Loop Response (Bode) measurement configuration menu. Use this menu to set the measurements source, add a custom name (label) for the measurement and other parameters.

Field or control	Description
Input Source	Selects the channel connected to the input side of the DUT. Tap the field to show the list of available input sources.
Output Source	Selects the channel connected to the output side of the DUT. Tap the field to show the list of available output sources.

Field or control	Description
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Impedance	A user configurable Impedance for the generator. Sets the impedance as **50 Ω** or **High Z**.
Start Frequency	Sets the generator start frequency. The default start frequency is 100 Hz. Refer to Supported frequency range for the Generator. Double-tap in the field and change the start frequency using a connected keyboard or the virtual keyboard.
Stop Frequency	Sets the generator stop frequency. The default stop frequency is 20 MHz. Refer to Supported frequency range for the Generator. Double-tap in the field and change the stop frequency using a connected keyboard or the virtual keyboard.
Amplitude Mode	Sets the built-in signal generator output amplitude mode. **Constant** Sets the constant amplitude for all frequencies in the internal signal generator. **Profile** Opens a configuration menu to set the amplitude for each frequency band. The generator uses the amplitude values to set the output amplitude as the frequency changes. Refer Configure Profile table for amplitude values.
Configure Profile	Opens the profile configuration menu. See Configure Profile. Available when **Amplitude Mode** is **Profile.**
Results Table	Opens and displays the FRA Results Table, which contains the measurement result.
Amplitude	Sets the generator to apply the same amplitude to all frequencies output from the generator. The minimum value is 20 mV for the internal generator. Refer Configure Profile table for amplitude values. The default value is 100 mV. Available when **Amplitude Mode** is **Constant**.
Points Per Decade	Sets the number of frequency points between the start and stop frequency in terms of log scale. The minimum and default value is 10 frequency points. The maximum value is 400.
Preset	Sets the built-in generator such as start frequency, impedance, and amplitude values. It sets the optimal vertical, horizontal and trigger settings of oscilloscope. Measurement automation begins on clicking Preset.

NOTE: Math sources are not supported for Control Loop Response (Bode) measurement.

FRA from the Meas button

The following options are available when you tap on the Meas button and select the FRA tab.

Controls	Description
Input source	Input source of the control loop.
Output source	Selects the output source of the control loop.

Configure Profile for Frequency Response Analysis (FRA) measurements

Use the **Configure Profile** menu to configure the amplitude profile for the Control Loop Response (Bode) measurement.

It is recommended to customize the internal signal generator amplitude values using the amplitude profile configuration. Profiling allows you to set the generator to a lower amplitudes, at different frequencies where the DUT is sensitive, and set the higher amplitudes where the DUT is less sensitive to distortion. A Switched-mode Power Supply (SMPS) is very sensitive near the zero degree cross over frequency, so it is advisable to define a profile, instead of using a constant amplitude output signal.

The following table list the available configurations.

Field or control	Description
Step	Displays each step.
Start Frequency	Displays the start frequency.
Stop Frequency	Displays the stop frequency.
Amplitude	Shows the amplitude value used to set the AFG level.
Insert Step	Inserts an additional row displaying start, stop, and amplitude values.
Delete Step	Deletes the selected row.
Clear Table	Clears all the values in the table.

Use the following steps to configure the amplitude profile for a Control Loop Response (Bode) measurement:

1. Tap **Configure Profile** button. The configure profile dialog is displayed. The configure profile dialog displays two rows:
   - The start frequency in the first row is the start frequency from the Control Loop Response (Bode) measurement configuration panel.
   - The stop frequency in the first row is the mean of start and stop frequency values from the Control Loop Response (Bode) measurement configuration panel.
   - The start frequency in the second row is the same stop frequency value in the first row from the table of configure profile. This applies to all the following rows added in the table of configure profile.
   - The stop frequency in the second row is the stop frequency from the Control Loop Response (Bode) measurement configuration panel.
2. Edit the values as necessary to reduce the variations in the gain curve at particular frequencies where the DUT is sensitive to distortion. Adjust by making configuration changes and viewing the response in the gain/phase curve of the Control Loop Response (Bode) plot.
3. Tap outside the configure profile table save your updates.

Sample amplitude values for a DC-DC convertor DUT

Start	Stop	Amplitude
10 Hz	100 Hz	1 V
100 Hz	500 Hz	800 mV
500 Hz	1 kHz	600 mV
1 kHz	10 kHz	400 mV
10 kHz	100 kHz	350 mV
100 kHz	1 MHz	300 mV
1 MHz	10 MHz	250 mV
10 MHz	20 MHz	200 mV

NOTE: The maximum stop frequency value for the internal signal generator is 50 MHz.

Measurement configuration menu overview

Use this configuration menu to add statistics to a measurement badge readout and change measurement settings including source, reference levels, and gating.

To open a Measurement configuration menu for a measurement, double-tap a Measurement badge in the Results bar. The configuration menu and panels only show fields and controls relevant to the selected measurement.

The menu opens on the measurement name panel (the name of the measurement), which provides controls to display additional statistics to the measurement badge. The content of the measurement name panel depends on the measurement. The most common Measurement Name fields are listed in the following table.

Measurement configuration menu fields, controls, and panels

Field, control, or panel	Description
Measurement Statistics (Measurement name panel)	A list of measurement statistics related to the measurement. You can add these statistics to a measurement badge by selecting the **Show Statistics in Badge** control.
Show Statistics in Badge (Measurement name panel)	Adds the listed statistical measurement readouts to the measurement badge readout.
Results Table	Button that opens the Measurement Results of the measurement. The results table displays all the statistics of the measurement. To add a results table to the screen, double-tap the measurement badge and tap **Results Table**. See Add Results Table configuration menu.

Field, control, or panel	Description
Configure panel	Sets the source, label text, and other fields that are specific to each measurement type. See Configure panel (Measurement configuration menu).
Reference Levels panel	Sets the reference levels and units used to take measurements and the method used to calculate the Top and Base waveform values. See Reference Levels panel (Measurement configuration menu).
Gating panel	Sets the measurement region (gate) used to take measurements. Select the type of gating to use. See Gating panel (Measurement configuration menu).

Measurement Name panel (Measurement configuration menu)

The Measurement Name panel (the name of the measurement) provides controls for adding display statistics to the measurement badge and opening Results Table of the measurement.

To open the measurement name panel, double-tap a Measurement badge. This is the default panel shown when you open a Measurement settings menu.

The content of the Measurement Name panel depends on the measurement.

Field or control	Description
Measurement Statistics	A list of measurement statistics. You can add these statistics to a measurement badge by selecting the **Show Statistics in Badge** control.
Show Statistics in Badge	Select to add the listed statistical measurement readouts to the measurement badge.
Results Table	Adds the Measurement Results of the measurement to the screen. The results table displays all the statistics of the measurement.

See also

Configure panel (Measurement configuration menu)

Reference Levels panel (Measurement configuration menu)

Gating panel (Measurement configuration menu)

Configure panel (Measurement configuration menu)

Use the Configure panel to set the measurement source(s), add a custom name (label) for the measurement, and other parameters.

To open the Configure panel:

1. Double-tap a Measurement badge to open the Measurement configuration menu.
2. Tap the **Configure** panel.

Not all items listed are shown for all measurements; The panel only shows fields and controls relevant to the selected measurement.

Field or control	Description
Source	Sets the signal source used to take the measurement. Tap the field to show the list of available sources. If the measurement requires more than one source, multiple Source fields are displayed.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

Other measurement panels

Measurement Name panel (Measurement configuration menu)

Reference Levels panel (Measurement configuration menu)

Gating panel (Measurement configuration menu)

Reference Levels panel (Measurement configuration menu)

Use the Reference Levels panel to set the reference levels (High, Mid, and Low), the units used to take measurements, and the method used to calculate the Top and Base waveform values. You can set the levels to be the same or different for rising and falling edges.

To open the Reference Levels panel:

1. Double-tap a Measurement badge.
2. Tap the **Reference Levels** panel.

Reference Levels panel- fields and controls

Not all items listed in the table are shown for all measurements; The panel only shows fields and controls relevant to the selected measurement.

Field or control	Description
Reference Levels	All measurement reference levels are per source.
Source	Lists the source signals used for each edge of the measurement.
Set Levels In	Sets the method used to set or calculate the High, Mid, and Low reference levels. Select % or Units and use the Multipurpose Knob to set custom reference values.
Levels	Sets the reference levels as specified percentages of the Top and Base waveform measurement. To set custom reference values, tap **Custom**, tap a setting field, and use the Multipurpose Knob to set the different % (relative) or absolute values. **High** and **Low** references are used to calculate rise and fall times. The default High reference is 90% and Low reference is 10%. **Mid** reference is primarily used for measurements between edges such as pulse widths. The default level is 50%.

Field or control	Description
Base Top Method	Sets the method to calculate the waveform Base and Top values, which is then used to calculate the High, Mid, and Low reference levels. **Auto** is the default method, which automatically determines the most suitable Base Top method to use. It initially applies the Histogram Mode method. If the Mode value for either the top or bottom half of the waveform is unstable or ambiguous, the MinMax method is used instead. **MinMax** Uses the minimum and maximum values in the waveform record to determine the base and top amplitude. Useful on a waveform with low noise and free from excessive overshoot. **Histogram Mode** uses histogram analysis to calculate the most common values above and below the waveform midpoint. Top is set to the common high value, and Base is set to the common low value.
Hysteresis	Sets the threshold margin to the reference level which the signal must cross to be recognized as changing; the margin is the relative reference level plus or minus half the hysteresis. Use hysteresis to filter out spurious events. Tap the field and use the Multipurpose Knob to change the value.

Other measurement settings panels

Measurement Name panel (Measurement configuration menu)

Configure panel (Measurement configuration menu)

Gating panel (Measurement configuration menu)

Gating panel (Measurement configuration menu)

Use Gating to confine a measurement to a certain part of a waveform.

To open the **Gating** panel:

1. Double-tap a Measurement badge in the Results bar to open the Measurement configuration menu.
2. Tap the **Gating** panel.

Gating panel, fields and controls

Not all items listed in the table are shown for all measurements; The panel shows fields and controls relevant to the selected measurement.

Field or control	Description
Gating Type	Sets the gate type used to take measurements. **None**: Measurements are taken across the entire record. **Screen**: Measurements are taken on the portion of the waveform shown on the display. When zoom is on, the ‘display’ on which to measure is the zoom window. **Cursors**: Measurements are taken on the portion of the waveform between the cursors.

Guidelines

- Only one gated region is allowed per measurement. You can create duplicate measurements and define different gates for each one.
- Tap on a gating-enabled measurement badge to display the vertical gate bars on the waveform for that measurement.
- Gating is indicated by two vertical bars; one at the start of the gated region and one at the end.
- If the measurement has more than one source, gate bars are drawn on both sources.

See also

Measurement Name panel (Measurement configuration menu)

Configure panel (Measurement configuration menu)

Reference Levels panel (Measurement configuration menu)

Add Plot configuration menu

Use this configuration menu to select and plot the amplitude and time relationship of two (XY) signals, which can be sourced from channel, math, or reference waveforms.

To add a plot:

1. Double-tap **Horizontal** badge on the Setting bar.
2. Tap the **Acquisition Settings** panel.
3. Tap **XY** to **On**. The plot is added to the screen.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

Add Results Table configuration menu

Use this configuration menu to add a table of all active measurements, searches, or bus decode values. Results tables show values in a spreadsheet-like format.

To add a Measurement result table to the screen:

1. Double-tap a measurement badge.
2. Set the Results Table to **On**. The Measurement result table is added to the screen.

To add a Bus Decode Results table to the screen:

1. Double-tap a bus badge.
2. Set the Results Table to **On**. The Bus Decode Results table is added to the screen.

Add Results Table menu fields and controls

Field or control	Description
Measurements	Display a table of all the measurements in the Results bar.
Bus Decode	Display a table of the bus decode results.

Results Tables operations overview

Results tables list summaries of all active measurements, bus decode activity, and search results in a spreadsheet format. Use Results tables to quickly compare values or save the results to a report.

Results Table - general operations

- Double-tap anywhere on a results table to open its configuration menu.
- To scroll up and down in a results table, tap the table and swipe with your finger to scroll.
- To add or remove a column from the table, double-tap the results table to open the table configuration menu and select from available columns to add or remove.
- You cannot sort the contents of a column.

See also

Measurement Results Table configuration menu

Bus Decode Results table configuration menu

Frequency Response Analysis (FRA) Results table configuration menu (on page 79)

Harmonics/Frequency Response Analysis (FRA) Results table configuration menu

Use this configuration menu to select the information to be displayed in the FRA Results Table, or save a Custom table to a file.

To open the FRA Results Table, double-tap the Power badge and set the Results Table to On.

To open the FRA Results Table configuration menu, double-tap anywhere in the FRA Results table.

Custom Results Table configuration menu

Field or control	Description
Table Settings panel	
Column Visibility	Select or clear individual check boxes to add or remove that column from the table.
Save panel	
Save Table	Opens a menu to save the Custom Results Table data to a comma-separated value (CSV) file. See Save As configuration menu, Custom Results Table.

Save As configuration menu, Custom Results Table

Use this configuration menu to set the location and file name to save the Custom Results Table content as a comma-separated value (CSV) file.

To open the Save As configuration menu to save a Custom Results Table:

1. Double-tap anywhere in the Custom Results Table to open the Custom Results Table configuration menu.
2. Tap the **Save** panel.
3. Tap the **Save Table** button.

Save As configuration menu, Custom Results Table

Field or control	Description
Save Location	Sets the location to save the file. The default value is the previous save location. Tap on the file path and enter the location. Use an external keyboard or double-tap on the file name to open the virtual keyboard and enter the path details. You can also use the **Browse** button to open a menu to navigate to and select the save location. Tap the down arrow icon on the right end of the field to open the list of recently saved file locations for the current save type. Up to 20 recently saved file locations are stored.
Browse	Tap to open the Browse Save As Location dialog, to navigate to and select the location to save the file. See Browse Save As Location configuration menu.
File Name	Enter the file name. The default file name is Tek000. The file name increments on the next save (Tek001, Tek002, and so on). Double-tap on the file name to open the virtual keyboard or use an external keyboard to enter the name.
Save as Type	You can only save a Custom Results Table to a comma-separated value (CSV) file.

Measurement Results Table configuration menu

Use this menu to select which statistics to show in the **Measurement Results** table, select the number of digits to show in each column, save a table, and so on.

To open the Measurement Results Table configuration menu, double-tap anywhere in a Measurement Results Table.

Measurement Results Table menu

Field or control	Description
Column Visibility panel	
Label	Adds a Label column to show the user-defined label for all measurements. If no user-defined label exists, the column shows the default measurement name.
Peak-to-Peak	Adds a Peak-to-Peak column and shows a Pk-Pk readout for all relevant measurements.
Check boxes to select columns to display	Select or clear individual statistic check boxes to add or remove that column from the table.
Column Resolution panel	
Measurement column resolution	Sets the resolution (number of digits to display) in each measurement column.
Set to Defaults	Sets all columns to show 5 digit readout resolution.
Other panel	

Field or control	Description
Statistics	Sets the amount of statistical information you want to display for each measurement. **Both** shows both the accumulated results and the current acquisition results. **All Acquisitions** shows statistics for all acquisitions from the last action that cleared acquisition memory. **Current Acquisition** shows the statistics for just the current acquisition.
Row Height Optimized For	Sets the row height of the Measurement results table as per your preference for **Mouse** or **Touch**.
Save Table	Opens a menu with which to save the results table data to a file. See Save As configuration menu (Measurement Results Table).

Measurements Results Table operations

- Double-tap anywhere on a results table to open its configuration menu.
- If you add or delete a measurement to the Results bar, that measurement is automatically added to or deleted from an existing Measurement Results table.
- To scroll up and down in a results table, tap the table and swipe with your finger to scroll.
- To add or remove a column from the table, double-tap the results table to open the table configuration menu and select from available columns to add or remove.
- You cannot sort the contents of a column.

Save As configuration menu (Measurement Results Table)

Use the Save As configuration menu to set the location and file name to which to save the contents of a results table, as a comma-separated value (csv) format file.

To open the Save As configuration menu to save a results table to a file:

1. Double-tap anywhere in the Measurement Results table to open the Measurement Table configuration menu.
2. Tap the **Other** panel.
3. Tap **Save Table**.

Save As menu, Measurement Results table fields and controls

Field or control	Description
Save Location	Sets the location to which to save the file. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, for the current save type, up to a maximum of 20 locations.
Browse	Tap to open the Browse Save As Location dialog, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu.

Field or control	Description
File Name	The file name assigned to the file. The default file name is Tek000. The file name increments on the next save (Tek001, Tek002, and so on). Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the on-screen keyboard and enter a file name.
Save as Type	You can save only table results as comma separated values (.csv) files.

Bus Decode Results table configuration menu

Use this configuration menu to select which information to show in each Bus Decode Results table, or save a Bus Decode Results table to a file.

To open the Bus Decode Table configuration menu, double-tap anywhere in the Bus Decode Results table.

If there are multiple bus decode results tables, each bus has a tab in the table. Tap the tab of the bus to display and/or configure, and then double-tap anywhere in that table to open the configuration menu for that table.

Bus Decode Results table menu

Field or control	Description
Column Visibility panel	
Check boxes for columns in table to display	Select or clear individual check boxes to add or remove that column from the table. The listed columns depend on the selected bus table in the Bus Decode Results table. Select the tab of the bus, at the top of the table, to which you want to make changes, then double-tap the table to open the configuration menu for that table. Available check boxes are ▪ Parity (h) ▪ Sync (h) ▪ Unframed (h) ▪ Packet Header (h) ▪ Error ▪ Packet Trailer (h)
Other panel	
Save Table	Opens the Save As dialog to let you save the table data to a comma-separated values (.csv) file. See Save As configuration menu (Bus Decode Results table).

Bus Decode Results Table operations

- Double-tap anywhere on a results table to open the configuration menu for that tab.
- To scroll up and down in a results table, tap the table and swipe with your finger to scroll.
- To add or remove a column from the table, double-tap the results table to open the table configuration menu and select from available columns to add or remove.
- You cannot sort the contents of a column.

Save As configuration menu (Bus Decode Results table)

Use this configuration menu to set the location and file name to which to save the contents of a results table, as a comma-separated value (csv) format file.

To open the Save As configuration menu:

1. Double-tap anywhere in the Bus Decode Results table to open the Bus Decode Results configuration menu.
2. Tap the **Other** panel.
3. Tap **Save Table**.

Save As menu (Bus Decode Results table) fields and controls

Field or control	Description
Save Location	Sets the location to which to save the file. The default value is the last location to which a file of this type was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, for the current save type, up to a maximum of 20 locations.
Browse	Tap to open the Browse Save As Location configuration menu, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu.
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a default name created by the instrument if this file type has not previously been saved with a custom file name. The default file name uses the format Tek000. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save as Type	You can only save table results as comma separated values (.csv) files.

Bus configuration menu

Use the Bus menu to select the bus type to display, configure the input sources, and set how to display the bus on the screen.

To open the Bus configuration menu:

- For an existing bus, double-tap the **Bus** badge in the Settings bar.
- To add a new Bus badge on the Settings bar, tap the **Math Ref Bus** > **Bus** button. This adds the Bus badge to the Settings bar and opens the Bus configuration menu.

Bus configuration menu - fields and controls

Field or control	Description
Display	Toggles bus display **On** or **Off**.
Label	Toggles the label **On** or **Off**. The label is the name of the bus type.
Position	Sets the vertical position of the bus waveform. Default is 0 (center of graticule).
Set to 0	Sets the vertical position of the bus waveform to 0 (center graticule).

Field or control	Description
Results Table	Sets the results table as **On** or **Off** to see the active measurements, bus decode activity, and search results in a spreadsheet format. Default is off.
Bus Type	Select a bus from the drop down list (SPI, RS-232, I2C).
Source configuration	A set of fields and controls that set the bus signal input parameters. Shown fields depend on the selected bus type. See the individual bus configuration help topics for information on their settings.
Decode format	Sets how decoded data information is shown in the bus. Select from listed formats. Available formats depend on the bus type.

Bus type configuration menus

Use the following links to access information on specific Bus configuration menus.

I2C serial bus configuration menu (on page 84)

RS232 serial bus configuration menu (on page 86)

SPI serial bus configuration menu (on page 85)

I2C serial bus configuration menu

Use the I2C bus menu to set up and display an I^2C (Inter-Integrated Circuit) serial bus waveform.

To set up the I^2C serial bus menu:

- To create a new I^2C bus waveform, tap the **Math Ref Bus** > **Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **I2C**.
- To change the settings on an existing I^2C serial bus waveform, double-tap the I^2C **Bus** waveform badge and make necessary changes in the configuration menu.

I2C serial bus menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	Toggles the label **On** or **Off**. The label is the name of the bus type.
Position	Sets the vertical position of the bus waveform.
Set to 0	Sets the vertical position to 0 divisions (centered vertically).
Results Table	Sets the results table as **On** or **Off** to see the active measurements, bus decode activity, and search results in a spreadsheet format. Default is **Off**.
Bus Type	Set to **I2C**.
SCLK Input	Sets the source and threshold level for the Serial Clock Line signal.
SDA Input	Sets the source and threshold level for the Serial Data signal.

Field or control	Description
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

SPI serial bus configuration menu

Use the SPI bus menu to set up and display an SPI (Serial Peripheral Interface) synchronous serial bus waveform.

To set up the SPI serial bus:

1. To create a new SPI bus waveform, tap the **Math Ref Bus** > **Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **SPI**.
2. To change the settings on an existing SPI serial bus waveform, double-tap the **Bus** waveform badge to open the configuration menu, and make necessary changes.

SPI serial bus menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	Toggles the label **On** or **Off**. The label is the name of the bus type
Position	Sets the vertical position of the bus waveform. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically).
Results Table	Sets the results table as **On** or **Off** to see the active measurements, bus decode activity, and search results in a spreadsheet format. Default is off.
Bus Type	Set to **SPI**.
Framing	Set to Slave Select (SS) or Idle framing modes.
SCLK Input	Select the channel source and threshold level for the Serial Clock signal (output from master). Set the Polarity to rising or falling edge of the clock signal used by the master device to start transferring bits.
SS Input	Select the channel source and threshold level for the Slave Select signal to start communications with the slave device. Set the Polarity to use Active High or Active Low logic for the SS signal. Available when **Framing** is **SS**.
Data Input	Select the channel source and threshold level for the data input. Set the Polarity to use Active High or Active Low for the SS signal.
Word Size	Enter the word size, in bits. Minimum is 4, maximum is 32, and default is 8.
Bit Order	Set to most significant bit **MS First** or least significant bit **LS First**.

Field or control	Description
Idle Time (Framing =)	Set the idle frame time. Available when **Framing** is **Idle**.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex** and **Binary**.

RS232 serial bus menu

Use this menu to set up and display an RS232 serial bus waveform.

To set up a RS232 serial bus:

1. To create a new RS232 bus waveform, tap the **Math Ref Bus** > **Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **RS232**.
2. To change the settings on an existing RS232 serial bus waveform, double-tap the RS232 **Bus** waveform badge to open the configuration menu and make necessary changes.

RS232 serial bus menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	Toggles the label **On** or **Off**. The label is the name of the bus type
Position	Sets the vertical position of the bus waveform. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically).
Results Table	Sets the results table as **On** or **Off** to see the active measurements, bus decode activity, and search results in a spreadsheet format. Default is **Off**.
Bus Type	Set to **RS232**.
Bit Rate	Sets the data bit rate.
Data Bits	Set the number of bits that define a data packet for your RS232 bus.
Source	Set the signal source from available analog channels.
Threshold	Set the threshold level to define a logic high level.
Polarity	Select the polarity to match the RS232 bus being acquired. Use Normal polarity for RS-232 signals, and Inverted polarity for RS-422, RS-485, and UART buses.
Parity	Set the parity to match the RS232 bus being acquired.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and ASCII.

Analog Channel configuration menu

Use the Analog Channel configuration menu to set up analog channel vertical settings, probe settings, external attenuation, alternate units for analog channel inputs for the channel.

To open an analog Channel configuration menu, double-tap an analog Channel badge. The following text describes analog channel settings.

Vertical Settings panel, fields, and controls

Field or control	Description
Display	Enables **(On)** or disables **(Off)** displaying the channel waveform in the Waveform View window. Disabling the channel waveform will prevent measurements from updating for the specified channel.
Invert	Reverses the amplitude polarity of each sample point. When On, Invert only inverts the displayed waveform and waveform data. It will not invert the bus decode data.
Vertical Scale	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Offset	Sets the channel signal vertical offset. When there is an offset, the channel handle displays it in the waveform view.
Set to 0	Sets the channel signal vertical offset to 0 units.
Position	Double-tap the field to set the vertical position using the virtual keypad.
Set to 0	Tap to set the waveform zero volt level to the center of the waveform view.
Label	Double-tap the field to add a label to the channel display using the virtual keypad.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Other channel settings panels	
Probe Setup	See Probe Setup panel Analog channel configuration menu (on page 87).

Probe Setup panel (Analog channel configuration menu)

Use the Channel configuration menu Probe Setup panel to setup probe, deskew.

To open the Probe Setup panel:

1. Double-tap an analog Channel badge on the Settings bar to open the Channel configuration menu.
2. Tap the Probe Setup panel.

Probe Setup panel fields and controls

Available fields and controls vary with the type of probe that is attached. For more information, consult the probe documentation.

Field or control	Description
Probe Type	Sets the probe type as **Voltage** or **Current**.
Attenuation	Set attenuation as **1X**, **10X**, or **Custom**.
Custom Attenuation	Set the value for Custom Attenuation. Available when **Attenuation** is **Custom**.
Measure Current	Measures the current. Available when **Probe Type** is **Voltage**.
Ratio	Double-tap the numeric fields to set the ratio using the virtual keypad. Available when **Probe Type** is **Voltage** and **Measure Current** is **Yes**.
Set to Unity	Sets the ratio to unity. Available when **Probe Type** is **Voltage** and **Measure Current** is **Yes**.

Invert channel guidelines

This topic describes how the channel Invert function interacts with other instrument settings.

Settings that change when Invert is set to On or Off:

- The instrument clears acquisitions and restarts triggering
- All persistence information resets/clears, including all waveform statistics

Settings that change when Invert is set to On:

- The instrument adds a down arrow to the title row of the channel badge to indicate inverted mode
- The channel **Offset** value reverses polarity
- The ADC value reverses polarity
- The trigger level value reverses polarity
- The trigger level indicator moves to the inverted value and the trigger level bar appears momentarily to show the new level
- The trigger **Slope** indicator on the Trigger badge and in its menu changes to the opposite edge (Rising to Falling, or Falling to Rising). If the Either edge mode is selected then slope mode does not change

Settings that don't change when Invert is On:

- The channel **Vertical** scale does not change
- The channel **Vertical** position knob rotation direction does not change
- The channel Vertical **Position** value does not change
- The **Trigger** level knob rotation direction does not change
- Bus decode is not supported with Invert enabled; if a waveform is inverted with a Bus decode on, the decode will be performed on the uninverted waveform

Invert and Setup, Autoset, and Default functions

- The invert state is saved and recalled correctly as part of **Setup** and **Session** files
- **Autoset** does not affect **Invert**
- **Default** sets **Invert** to **Off**

AFG configuration menu

Use the AFG configuration menu to set the output signal parameters for the optional arbitrary function generator. Use the AFG to simulate signals within a design or add noise to signals to perform margin testing.

To open the AFG configuration menu:

1. Tap the **DVM AFG** > **AFG** on the Settings bar. The oscilloscope shows the AFG configuration menu. You can change the **AFG** settings while the output is **Off**.
2. Double-tap the AFG badge to open the AFG menu.

Arbitrary Function Generator overview

The function generator provides output of predefined waveforms up to 25 MHz. Choose between Sine, Square, Pulse, Ramp, DC, Noise, and Arbitrary signals.

You can also select a predefined waveform or load a saved .wfm- or .csv-format waveform from storage.

Arbitrary Function Generator menu fields and controls

Not all items listed in the table may be shown for all measurements. The configuration menu only shows fields and controls relevant to the selected Waveform Type.

The output connector is on the rear panel, labeled **AFG**.

Field or control	Description
Output	Tap to toggle between the output **Off** or **On**.
Waveform Type	Tap to select an available waveform from the list.
Load	Maximum waveform size is 8192 points, and be set to a frequency of 1 µHz to 5 MHz. Any excess points will be downsampled. Available when **Waveform Type** is **Arbitrary**.
Waveform File	Shows the loaded waveform file path and name. Tap to select a waveform file to load into the AFG waveform memory from the drop-down list of the last 20 waveforms that have been loaded using the **Load** button.
Symmetry	Sets the symmetry of the ramp using the keypad or multipurpose knob. Available when the **Waveform Type** is **Ramp**.
Width	Sets the width of the pulse using the keypad or multipurpose knob. Available when the **Waveform Type** is **Pulse**.
Duty Cycle	Sets the duty cycle of the square wave using the keypad or multipurpose knob. Available when the **Waveform Type** is **Square**.
Frequency	Sets the frequency of the waveform using the keypad or multipurpose knob. The frequency range is 0.1 Hz to 25 MHz, in increments of 0.1 Hz.
Period	Sets the period of the waveform using the keypad or multipurpose knob.
Amplitude	Sets the amplitude of the waveform using the keypad or multipurpose knob

Field or control	Description
Offset	ets the offset of the waveform using the keypad or multipurpose knob.
High Level	Sets the High signal amplitude of the waveform using the keypad or multipurpose knob
Low Level	Sets the low signal amplitude of the waveform using the keypad or multipurpose knob.
Load Impedance	Tap to select either **50 Ω** or **High Z** (1 MΩ) output load impedance.
Add Noise	Tap the check box to toggle noise on and off. Set the amount of noise to add to the output signal using the keypad or the multipurpose controls.

Modulation menu fields and controls

NOTE: The modulation menu is not available when the Waveform Type is set to DC, Pulse, or Noise.

Field or control	Description
Output	Tap to toggle between modulation **On** or **Off**.
Modulation	Select one of the supported modulations. The default modulation is AM.
ModulationType	Tap to select an available waveform from the list.
Modulation Shape	Select the modulation shape from the drop-down list. ▪ Sine ▪ Square ▪ Up Ramp ▪ Down Ramp ▪ Arbitrary ▪ Noise Only shown when the Modulation Type = AM or FM
Modulation Frequency	Sets the frequency of the modulation using the keypad or multipurpose knob.
Depth	Sets the depth of the modulation using the keypad or multipurpose knob. Only shown when the Modulation Type = AM.
Frequency Deviation	Sets the deviation of the modulation using the keypad or multipurpose knob. Only shown when the Modulation Type = FM.
Hop Frequency	Set the hop frequency using keypad or multipurpose knob. Only shown when the Modulation Type = FSK.
Rate	Set the Hop Frequency Rate using the keypad or multipurpose knob. Only shown when the Modulation Type = FSK.

Field or control	Description
Type	Toggle between Linear or Log. Only shown when the Modulation Type = Sweep.
Start Frequency	Set the frequency of the waveform using the keypad or multipurpose knob. Only shown when the Modulation Type = Sweep.
Stop Frequency	Set the period of the waveform using the keypad or multipurpose knob. Only shown when the Modulation Type = Sweep.
Time	Set the time duration using keypad or multipurpose knob. Only shown when the Modulation Type = Sweep.

AFG output amplitude ranges by impedance (peak to peak)

Waveform	50 Ω	High Z (1 MΩ)
Sine	10 mV to 2.5 V	20 mV to 5 V
Square	10 mV to 2.5 V	20 mV to 5 V
Pulse	10 mV to 2.5 V	20 mV to 5 V
Ramp	10 mV to 2.5 V	20 mV to 5 V
DC Level	0 V to ±1.25 V	0 V to ±2.5 V
Random Noise	10 mV to 2.5 V	20 mV to 5 V
Arbitrary	10 mV to 2.5 V	20 mV to 5 V

Cursor configuration menu

Cursors are on-screen lines (bars) that you position in a Waveform or Plot views to take manual measurements on signals. They appear as horizontal and/or as vertical lines.

To open the Cursors configuration menu:

1. Double-tap a cursor readout or cursor line, or

To display cursors on the screen:

1. Tap the **Cursors** button on the display.

Cursor configuration menu fields and controls

Some fields or controls are available only when certain other controls are selected.

Field or control	Description
Display	Tap to toggle the cursor display **On** or **Off**.

Field or control	Description
Readout Location	Tap to set where to display cursor readouts. **Graticule** displays the cursor readouts on the screen cursor bars (default method). You can move the readouts on cursors by touching and dragging them along the cursor bar. **Badge** removes the screen cursor readouts and displays the cursor information in a **Cursors** badge in the Results bar. The **Cursors** badge content changes as you change the cursor type.
Cursor Type	Tap to select the cursor type from the drop-down list. **Waveform** cursors measure vertical amplitude and horizontal time parameters simultaneously at the point the cursor intersects a waveform. The cursor intersect point tracks waveform amplitude changes. **V Bars** are vertical cursors that measure horizontal parameters (typically time). They are not associated with the waveform, but show the time position of the cursor in the waveform record. **H Bars** are horizontal cursors that measure amplitude (typically in volts or amperes). They are not associated with the waveform, but show the amplitude position of the cursor. **V&H Bars** cursors measure vertical and horizontal parameters simultaneously. They are not associated with the waveform, but show the time and amplitude position of the cursors.
Source	Tap to select the source waveform from the drop-down list. **Selected Waveform** automatically moves the waveform cursors to the selected source or select a specific source from the drop-down list. **All** is available in **Source** drop-down list only when the **Cursor Type** is set to **Waveform**. **All** waveforms includes time domain based math waveforms, reference waveforms, trend waveforms, etc.
Cursor A X-Position	Tap to set a specific x-axis position for Cursor A using the multipurpose knob, or double-tap to set the position using the keypad.
Cursor B X-Position	Tap to set a specific x-axis position for Cursor B using the multipurpose knob, or double-tap to set the X-Position using the keypad.
Cursor A Y-Position	Tap to set a specific Y-axis position for Cursor A using the multipurpose knob, or double-tap to set the position using the keypad. Available when **Cursor Type** is **H Bars** or **V&H Bars**.
Cursor B Y-Position	Tap to set a specific Y-axis position for Cursor B using the multipurpose knob, or double-tap to set the X-Position using the keypad. Available when **Cursor Type** is **H Bars** or **V&H Bars**.

Field or control	Description
Cursor Mode	Tap to select the cursor mode. **Independent** mode sets multipurpose knobs A and B to move each cursor separately. **Linked** mode sets multipurpose knob A to move both cursors at the same time. Knob B will still move cursor B independently of knob A.

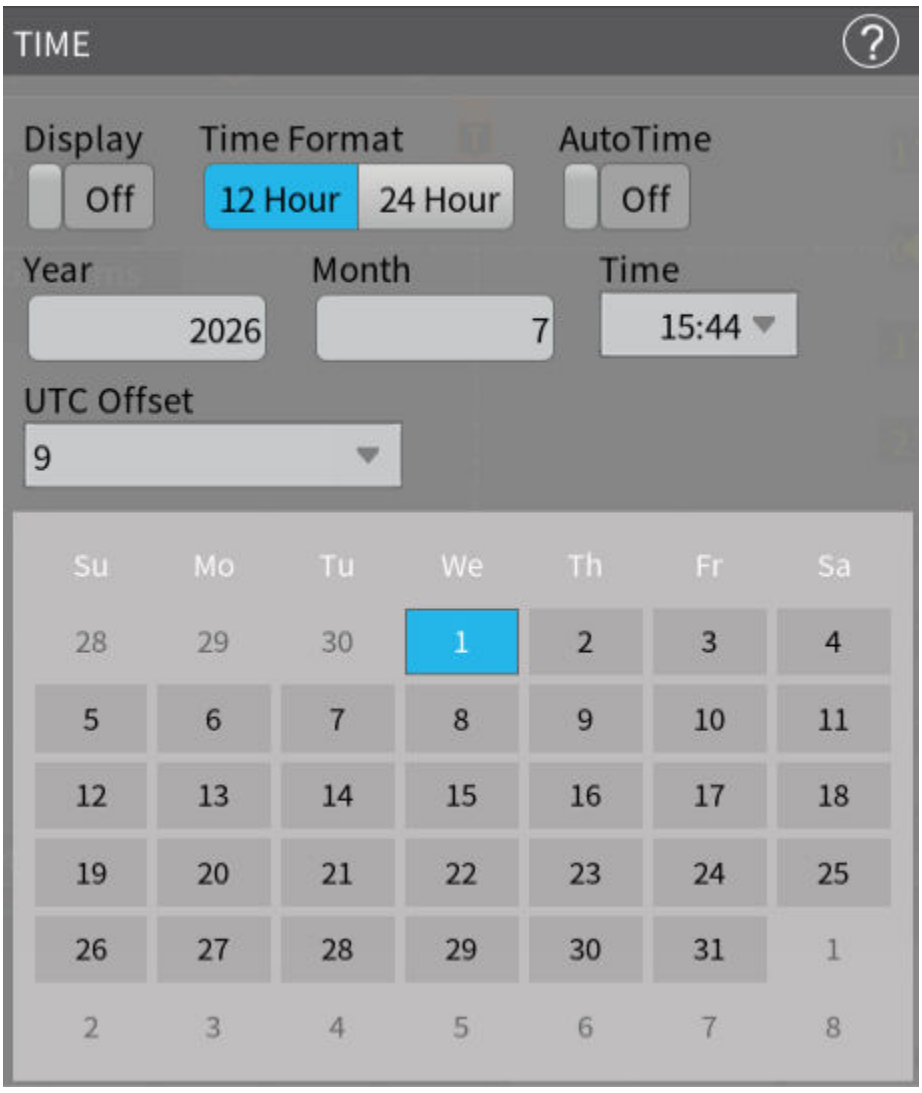
Date and Time configuration menu

Use this menu to set the date, time format, and time zone.

To open the Date and Time configuration menu, double-tap on the Date/Time badge in the lower-right corner of the oscilloscope display.

Date and Time configuration menu fields and controls

Field or control	Description
Display	Tap to toggle display of the date and time On or Off. When turned off, double-tap on the blank area below the Run/Stop button, in the lower-right corner of the oscilloscope display, to open the configuration menu and set display to On.
Time Format	Tap to select either 12 Hour or 24 Hour time format.
Year	Set the year using the keypad or multipurpose knob.
Month	Set the month using the keypad or multipurpose knob.
Time	Set the time using the keypad or multipurpose knob.
UTC Offset	Set the UTC offset based on the applicable time zone.
Date	Tap on the day in the calendar to set the date.
Autotime	Tap to toggle AutoTime On or Off. When enabled and connected to the internet, AutoTime will set with current time based on current UTC offset settings.



DVM configuration menu

Use this menu to set up the digital voltmeter (DVM) function to use probes to measure AC, DC, or AC+DC voltages. The DVM can also act as a frequency counter to measure frequencies on the analog input channels.

To open the digital voltmeter menu:

1. If the DVM is Off, tap the **DVM/AFG** badge and then select DVM. This adds a DVM badge to the top of the Results Bar, using the source that was selected the last time the DVM was added to the Results bar.
2. If the DVM is On, double-tap the **DVM** badge to open its configuration menu.

NOTE: Selecting a source in the DVM configuration menu does not automatically turn on (display) the source channel if the source channel is not already on.

DVM configuration menu fields and controls

Field or control	Description
Display	Tap to toggle the DVM badge On and Off.

Field or control	Description
Autorange	Tap to toggle autoranging On and Off. Autorange is not available when the oscilloscope is triggering on the same channel that is being measured.
Source	Tap to select the channel to measure from the drop-down list.
Mode	Tap to select measuring DC, AC RMS, or DC+AC RMS, or Frequency.
Show Basic Statistics in Badge	Tap to toggle showing DVM measurement statistics in the DVM badge.

Horizontal configuration menu

Use this menu to select the horizontal mode, set horizontal parameters, and enable trigger delay.

To open the **Horizontal** configuration menu, double-tap the **Horizontal** badge in the Settings bar.

Horizontal configuration menu fields and controls

Field or control	Description
Horizontal Scale	Tap to set the Horizontal Scale using the assigned multipurpose knob, double-tap to set the scale using the virtual keypad, or tap the up and down arrows. You can also use the front-panel Horizontal Scale knob to change this value. The horizontal scale determines the size of the acquisition window relative to the waveform. You can scale the window to contain a single waveform edge, a single cycle, several cycles, or thousands of cycles. The Sample Rate of the oscilloscope will automatically adjust when you set the horizontal scale.
Delay	Delay positions the trigger event to a specified time relative to the center of the waveform record. Use delay to focus on events that occur before (pre-trigger) or after the trigger point (post-trigger).
Position	Tap to set the trigger Position using the assigned multipurpose knob or double-tap to set the Position using the virtual keypad. When horizontal Delay is on, the time from the trigger point to the horizontal reference (center of waveform record) is the horizontal delay. The horizontal position determines the number of pre-trigger and post-trigger samples in the waveform record. When horizontal delay is off, the trigger point and the horizontal reference are at the same time in the middle of the waveform record.
Set to 0	Tap to set the delay position to 0 s (center of the waveform record). Available only when **Delay** is **On**.
Set to 10%	Tap to set the trigger delay to 10% of the waveform record. Available only when **Delay** is **Off**.

Field or control	Description
Record Length	Tap to set the Record Length using the assigned multipurpose knob, double-tap to set the length using the virtual keypad.
Run/Stop	Toggles the oscilloscope between constant acquisition (Run) and no acquisitions (Stop). When stopped, the oscilloscope shows waveforms from the last completed acquisition
Single/Seq	Acquires a single acquisition or a set number of acquisitions, then stops.
Clear	Erases acquired waveform data points from memory. Applies to all live acquisition waveforms.
Number of Waveforms	Sets the number of waveforms for the average result of several acquisitions. Available when **Acquisition Mode** is **Average**.
XY	Toggles the XY **On** or **Off** to plot the XY plot. Default is Off.
Single Sequence/Stop After	Enables stopping acquisitions after a specified number of acquisitions. Only works when using the Single/Sequence button.
Acquisition Mode	**Sample** creates a record point by saving one or more samples during each acquisition interval. Sample mode is the default acquisition mode. The instrument does no post processing of the acquired samples in this mode.
	High Res improves measurement accuracy by reducing random noise through sample averaging. When a sample rate lower than the maximum sample rate is selected, the oscilloscope combines multiple adjacent high-speed ADC samples into a single displayed sample. This process acts as a low-pass filter, reducing noise and producing a cleaner waveform. **NOTE:** The maximum vertical resolution of High Res mode is 12 bits.

DRAW A BOX Menu

Use this menu to toggle the Waveform view Zoom area draw box.

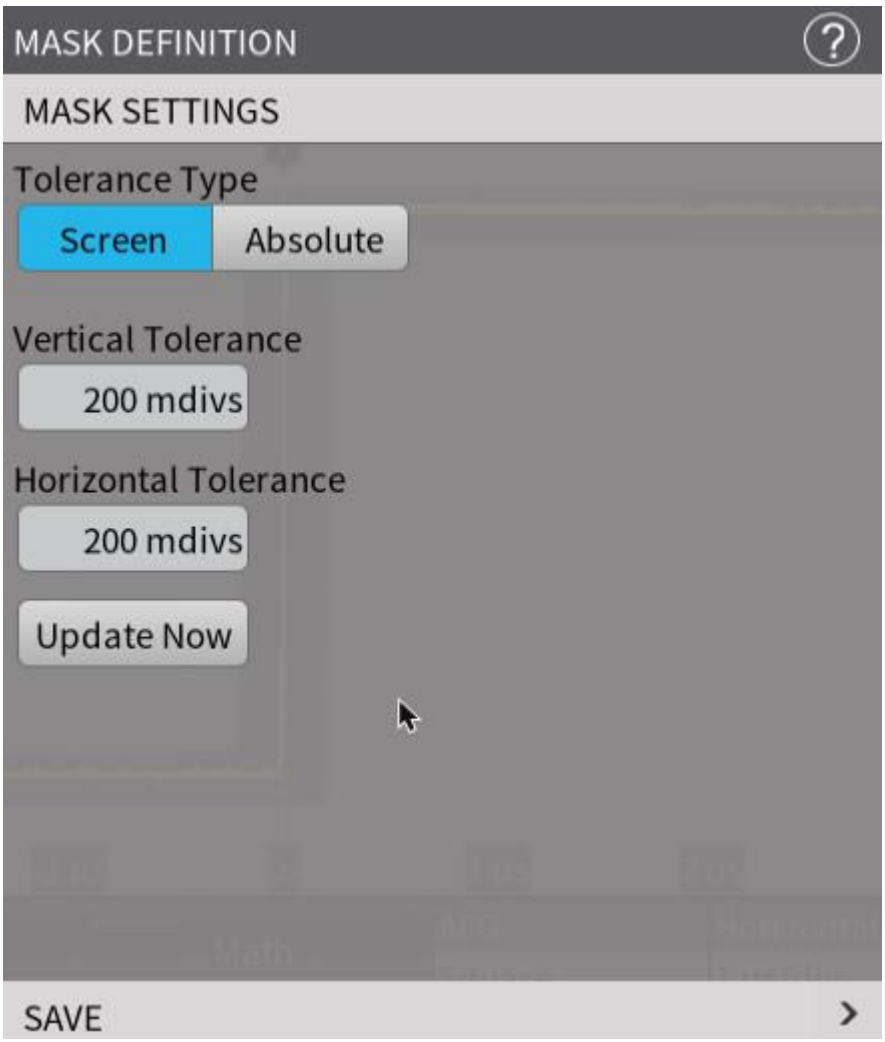
To use the **DRAW A BOX** menu, tap the **DRAW A BOX** window located at the second row of the left side of the Results Bar. This enables drawing a zoom box area on the screen.

Mask Definition Segment configuration menu

To open the **Mask Definition** menu, double-tap on a mask segment.

To create a mask segment, see Create a mask on waveforms (on page 59).

Waveform Mask Settings panel fields and controls



Field or control	Description
Tolerance Type	Set the Mask Tolerance as **Screen** or **Absolute**. Default type is Screen.
Vertical Tolerance	Tap the Vertical Tolerance field and enter the tolerance value as divisions of the current channel settings with **Screen** tolerance, or as volts in **Absolute** tolerance. Or double-tap on the field and use the A knob to change the value.
Horizontal Tolerance	Tap the Horizontal Tolerance field and enter the tolerance value as divisions of the current channel settings with **Screen** tolerance, or as seconds in **Absolute** tolerance. Or double-tap on the field and use the A knob to change the value.
Update Now	Updates the mask as per the configured tolerance values.

Mask Definition right click menu

See Right click menu functions associated with mask segments.

Mask Test badge configuration menu

A Mask Test badge is created when the first segment of a mask is defined. Use the Mask Test badge menu to configure mask test settings and define actions based on test results.

To open the **Mask Test** badge configuration menu, double-tap on the badge.

Mask Test badge menu

Field or control	Description
Mask Test	Sets the mask test as **Off** or **On**
Mask Display	Turns the mask display **On** (default) or **Off**. Turning off the mask display turns off the mask test also.
Label	The text field to add a label to the mask test badge. By default, this field is blank. When you recall a mask from a file, the name of the file is automatically displayed in the label field.
Source	Lists the valid source signals for mask testing. Valid sources do not include other sources that are already being used in mask tests.
Number of Waveforms	The numeric field to define the number of waveforms to test against. This value is shared amongst all mask tests. Changing the value in one badge changes it in all mask test badges. Maximum number of waveforms that can be tested is 4G. This field can be used along with the **Single / Seq Stops After** control to run a mask test with 100 waveforms ten times.
Total Failures	The numeric field to set the failure threshold in number of acquisitions. The value must be greater than or equal to Consecutive Failures.
Consecutive Failures	The numeric field to set the failure threshold in number of consecutively failing acquisitions. The value must be less than or equal to Total Failures.

Mask Test badge menu

See Right click menu functions associated with the Mask segments

Right click menu functions associated with the Mask Test badge

The following functions are available when you right click on a mask test badge.

Field or control	Description
Configure Mask Test	Opens the Mask Test badge configuration menu for the selected badge.
Delete Mask Test	Deletes the selected mask test badge.

Math configuration menu overview

Math waveforms are created by combining and/or mathematically transforming source waveforms into a new waveform for analysis. Use this menu to create math waveforms (basic or advanced) or add an FFT (Fast Fourier Transform) waveform to the screen.

To access a Math configuration menu, double-tap a **Math** waveform badge on the Settings bar. If no Math badge is present, tap the **Math Ref Bus** > **Add New Math** button to add a Math waveform and open the configuration menu.

Use the following links to access information on the Math waveform menus and settings.

Math configuration menu

Equation Editor (Math configuration menu)

Math configuration menu

Use this menu to set math waveform parameters, create basic and advanced math waveforms, or add an FFT (Fast Fourier Transformation) plot to analyze frequency components of a waveform.

To access the Math menu, double-tap a **Math** waveform badge. If no Math badge is present, tap the **Math Ref Bus** > **Add New Math** button to add a math waveform and open the menu.

Math configuration menu fields and controls

Field or control	Description
Display	Turns the math waveform or FFT plot **On** or **Off**.
Vertical Scale	Sets the vertical graticule scale units. Tap the arrows to change the value, tap and use the assigned multipurpose knob to change values, or double-tap to open the virtual keypad to enter a specific value. Available when **Auto Scale** is disabled (unchecked) and **Math Type** is **Basic** or **Advanced**.
Auto Scale	Toggles Auto Scale mode on or off. Auto Scale calculates the vertical scale and position to center and display the entire waveform. Available when **Math Type** is **Basic** or **Advanced**.
Label	Enter a label for the math waveform.
Position	Sets the vertical position of the math waveform. Available when **Math Type** is **Basic** or **Advanced**.
Set to 0	Sets the vertical position of the math waveform to zero. Available when **Math Type** is **Basic** or **Advanced**.
Alternate Units	Sets user-specified units to display on vertical scale readouts and measurement badges.
Average	Toggles averaging the waveform **On** and **Off**. Use averaging to reduce noise on the waveform.
Number of Averages	Sets the number of waveform acquisitions to average. Tap and use the assigned multipurpose knob to set the value. Available when **Average** is **On**.

Field or control	Description
Math Type	Sets the type of math waveform to display. **Basic** creates a math waveform by adding, subtracting, multiplying, or dividing two analog or reference waveforms. **FFT** opens an FFT view of the specified signal to display the frequency components of that signal. Double-tap on the FFT view to open its configuration menu. **Advanced** displays a drop-down from which to select the 10 last-accessed equations created by the Equation Editor. This mode also provides access to the Equation Editor. See Equation Editor (Math configuration menu).
Source, Source1, Source 2	Defines the signal source or sources for a **Basic**, **FFT** math waveform. **NOTE:** Basic and FFT math waveforms are created from analog channels and reference sources only.
Basic math operation list	Located between the **Source 1** and **Source 2** fields. A drop-down list to select a basic math operation (add, subtract, multiply, divide) to apply to the two sources. Available when **Math Type** is **Basic**.
Math n =	Lists the last-accessed advanced equation. Tap the down arrow to display a list of the last-accessed equations (up to a maximum of 10) created by the **Equation Editor**. Select an equation to display that math waveform. Tap **Edit** to open the **Equation Editor** to edit the displayed equation. Available when **Math Type** is **Advanced**.
Edit	Opens the **Equation Editor** to create advanced math waveforms from analog channels, references sources, constants, and functions. Tap the **Edit** button to open the **Equation Editor**. See Equation Editor (Math configuration menu). Available when **Math Type** is **Advanced**.

Math waveform guidelines

- You can take measurements on math waveforms in the same way as on channel waveforms.
- Math waveforms derive their horizontal scale and position from the sources in their math expressions. Adjusting these controls for the source waveforms also adjusts the math waveform.
- You cannot Zoom on math waveforms.

Equation Editor (Math configuration menu)

Use the Equation Editor to build your advanced math waveform expression using sources, operators, constants, measurements, and functions.

To access the math Equation Editor:

1. Double-tap a **Math** waveform badge. If no Math badge is present, tap the **Math Ref Bus** > **Add New Math** on the Setting bar to add a Math waveform and open the configuration menu.
2. Set Math Type to **Advanced**.
3. Tap **Edit** to open the Equation Editor.

Equation Editor menu fields and controls

Field or control	Description
Sources	Lists all available sources that you can add to an equation. Tap a source icon to add it to the cursor position in the Math x = input box.
Functions	Select the math functions to apply to your signal or signals. See Add Functions (math Equation Editor) .
Keypad	Use to enter numeric, basic math operations, and logic conditions.
Math <x> =	The field that lists the equation components that you add with the equation controls for the numbered Math waveform.
Clear	Clears the math equation field.
Arrows	Use to move the cursor position in the **Math <x> =** equation field.
Space	Use to add the space to the right of the cursor.
Delete	Deletes the first element to the right of the cursor location.
Bksp	Backspaces from the current cursor position, deleting the character to the left of the cursor.
Apply	Tap to apply the math expression shown in the **Math <x> =** field and display the math waveform. The first ~ ten characters of the equation also appears in the Math badge on the Settings bar. When the editor detects a logic or syntax error with the equation, the editor displays an error message, along with the character position in the equation where the editor considers the error to have occurred.
Cancel	Closes the editor menu, does not update the equation list if you made any changes from the last time you Applied or OK'd an equation.
OK	Applies the equation to the math waveform, closed the Advanced Editor window, and adds the equation to the available equation list.

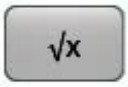
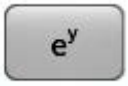
Equation editor guidelines

- Math definitions are not applied if the sources are not valid.
- Syntax or other equation or source errors display a short error message on the Math configuration menu.
- Use parentheses to group terms in the expression to control execution order, for example, 5*(Ch1 + Ch2).

Add Functions (math Equation Editor)

Use the Add Functions controls to add predefined math operations to your math waveform equation.

Button	Description
	Integral. Inserts the text INTG(into the math expression. Enter an argument to the function. The integral function produces the integral of the argument.

Button	Description
	Base 10 logarithm. Inserts the text LOG(into the math expression. Enter an argument to the function. The log function produces the base 10 logarithm of the argument.
	Square root. Inserts the text SQRT(into the math expression. Enter an argument to the function.
	Absolute. Inserts the text FABS(into the math expression. The FABS function takes the absolute value of the expression.
	Derivative. Inserts the text DIFF(into the math expression. Enter an argument to the function. The derivative function produces the derivative of the argument.
	Natural logarithm. Inserts the text LN(into the math expression. The natural logarithm function produces the natural logarithm of the argument.
	Natural antilog. Inserts the text EXP(into the math expression. The EXP function produces the natural antilog of the argument.
	FFT Magnitude. Inserts the text FftMag(into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT waveform that shows the magnitude components of the source signal.

Plot configuration menus

Use plot configuration menus to change settings of a displayed plot. Double-tap a plot view to open its configuration menu.

Plot XY configuration menu

Use this menu to change settings of a displayed XY plot.

To open the XY plot menu, double-tap anywhere in the **XY** Plot view.

Settings panel (Plot XY configuration menu) fields and controls

Field or control	Description
X-Axis	Sets the source for the X-axis signal.
Y-Axis	Sets the source for the Y-axis signal.

- Use the vertical scale and position controls of the waveform connected to the X- and Y-axis sources to set horizontal scale and position of the XY waveform.
- XY format is particularly useful for studying phase relationships between two similar signals, creating Lissajous Patterns.
- XY format is a dot-only display, although it can have persistence. The Vector style selection has no effect when you select XY format.
- XY format is not compatible with Average Mode.

Save panel fields and controls

Use the Save panel controls to save the plot image or date to a file for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the Save As menu. Navigate to the location where you want to save the plot image. Enter a file name in the File Name field. Select the image file format (PNG, BMP, or JPG). Tap **Save** to save the plot image.
Save Plot Data	Tap to open the Save As menu. Navigate to the location where you want to save the plot data. Enter a file name in the File Name field. Tap **Save** to save the plot data as a comma-separated values (.csv) file.

Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view.

Deleting the Measurement badge that enabled the plot also closes the plot.

Save As configuration menu (plot Save panel, Save Plot Image button)

Use this menu to specify the name and location at which to save an image file for the selected plot.

To access the **Save As** configuration menu to save an image of a plot to a file:

1. Double-tap anywhere in a Plot view top open the plot configuration menu.
2. Tap the **Save** panel.
3. Tap the **Save Plot Image** button.

Save As configuration menu (plot Save panel, Save Plot Image button) fields and controls

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the Browse button to select a save location. Double tap on the file name to open the virtual keyboard and enter a file name. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Opens the Browse Save As Location configuration menu to navigate to and select a location at which to save the file.

Field or control	Description
File name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Double-tap on the file name to open the virtual keyboard and enter a file name.
Save As Type	Lists the available graphic formats to which you can save files. Tap the field and select the graphic save format.
Cancel	Cancels the file save action and closes the configuration menu.
OK	Saves the file to the specified location, closes the Save As menu, and displays a confirmation message.

Save As configuration menu (plot Save panel, Save Plot Data button)

Use this menu to specify the name and location at which to save a comma separated value (csv) file of the data for a specific plot.

To access the Save As configuration menu for plots:

1. Double-tap anywhere in a Plot view top open the plot configuration menu.
2. Tap the **Save** panel.
3. Tap the **Save Plot Data** button.

Save As configuration menu (plot Save panel, Save Plot Data button) fields and controls

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the Browse button to select a save location. Double tap on the file name to open the virtual keyboard and enter a file name. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Opens the Browse Save As Location configuration menu to navigate to and select a location at which to save the file.
File name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Double-tap on the file name to open the virtual keyboard and enter a file name.
Cancel	Cancels the file save action and closes the configuration menu.
OK	Saves the file to the specified location, closes the Save As menu, and displays a confirmation message.

Control Loop Response (Bode) plot configuration menu

Use this menu to change the settings of a displayed Control Loop Response plot (Control Loop Response measurement).

To open the Control Loop Response plot configuration menu, double-tap anywhere in a **Control Loop Response** plot view.

Settings panel (Control Loop Response plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale On** or **Off**. Turn **AutoScale** off to manually set the X and Y-axis range and view an area of interest.
Grid lines	Selects which grid lines to show in the plot. Select the grid style that meets your measurement needs.
Display	**Vectors** draw waveforms with lines between record points. **Dots** draw waveform record points as dots on the screen.
Interpolation	Select the method to display the record points between sampled points. **Sin(x)/x** connects using a Sin(x)/x curve between the computed values. This makes the gain or phase curve smoother than linear interpolation. **Linear** connects between computed points (Gain or Phase value) using a straight-line fit. Measurement annotations are displayed for Linear mode interpolation only.
X-Axis (Frequency) Scale	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed only when **AutoScale** is off (deselected).
Y-Axis (Gain) Scale	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when **AutoScale** is off (deselected).
Y-Axis (Phase) Scale	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when **AutoScale** is off (deselected).

Save panel field and controls

Use the Save panel controls to save the plot image or date to a file for inclusion in reports or further analysis in other applications.

Field or control	Description
Save Plot Image	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot image. Enter a file name in the **File Name** field. Select the image file format (PNG, BMP, or JPG). Tap **Save** to save the plot image.
Save Plot Data	Tap to open the **Save As** menu. Navigate to the location where you want to save the plot data. Enter a file name in the **File Name** field. Tap **Save** to save the plot data as a comma-separated values (CSV) file. It saves the selected standard limits and computed Harmonic values.

Closing a plot view

To close (delete) a Plot view, tap the X in the upper right corner of the view.

Reference waveform configuration menu

Use this menu to configure display settings for a reference waveform.

To open a reference waveform configuration menu, double-tap a **Ref** badge on the Settings bar.

Reference waveform configuration menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the waveform.
Vertical Scale	Set the vertical scale by using the assigned multipurpose knob, a virtual keypad, or tap the up or down arrows.
Label	Adds a label to the waveform. Double-tap and enter text using the virtual keyboard. The label text is the same color as the waveform.
Units	Set the units label that you want to display on the vertical scale.
Position	Set the vertical position of the waveform using the assigned multipurpose knob or the virtual keypad.
Set to 0	Tap to set the vertical position to 0 (vertical center of the graticule).
Reference File	Shows the path and file name of the current Reference waveform. Tap the down arrow icon to list the 10 most recently accessed reference waveform files.
Browse	Opens the Browse Waveform File dialog. Use this dialog to navigate to and select a waveform file to load. (See Recall configuration menu (Ref waveform configuration menu).

Field or control	Description
Sample Rate, Record Length	Readout-only text that show the sample rate and record length values of the reference waveform.
Deskew	Changes the horizontal position of the reference waveform. Use this function to align the reference waveform to meet your measurement needs.
Set to 0	Sets the reference waveform deskew value to zero.

Recall configuration menu (Ref waveform configuration menu)

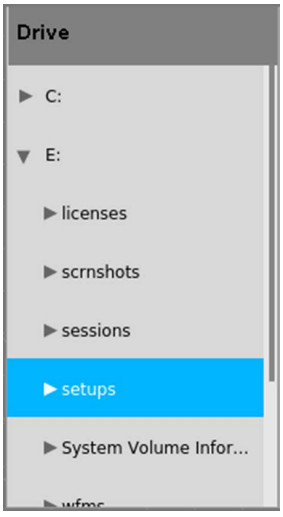
Use this menu to locate and load a reference waveform file.

Prerequisite: A Ref badge must be present on the Settings bar. See Add a math, reference, or bus waveform.

To open the Recall configuration menu:

1. Double-tap a **Ref** badge on the Settings bar.
2. Tap **Browse** to open the Recall configuration menu.

Recall configuration menu (Ref configuration menu) fields and controls

Field or control	Description
Look in	Shows the current directory path and file name. Select folder and file from the Drive field. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.
	The Drive column lists the directory structure, including any network-connected drives. Use to quickly navigate to a location. Tap to list the contents of the directory in the file list area. Double-tap an item, or tap the small arrow to the left of the directory, to display the subdirectories under it. Double-tap again to close that directory structure. If there is a scroll bar, drag the list up and down to show more entries.
	Use the arrow buttons to navigate the directory structure. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder. The Up arrow navigates up one level from the current folder.

Field or control	Description
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File Name	Lists the selected file name. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.
Files of Type	Use to select the file format you want to open. Tap the field to show a list of all file extension types that the instrument can read for the selected file type. The files column only lists files of the specified type.
Cancel	Cancels any changes, closes the menu, and returns to the prior menu without loading a file.
Open	Closes the dialog, returns to the prior menu, and loads and displays the specified file.

Trigger configuration menu overview

Use the Trigger menu to define the channel or waveform signal conditions on which to trigger the oscilloscope. The trigger event establishes the time-reference point in the waveform record. All waveform record data is located in time with respect to the trigger point.

To quickly change the trigger source from one channel to another, double-tap on the **Trigger** badge. Select the new trigger source from the list. If you select a Digital channel, the list expands to show the individual digital channels. Single source trigger types include **Edge**, **Pulse Width**, **Timeout**, **Runt**, and **Rise/Fall Time**.

Use the following links to see more information on specific trigger types and their configuration menus.

- Bus Trigger Menu
- Edge Trigger menu
- Logic Trigger
- Pulse Width Trigger menu
- Rise/Fall Time Trigger menu
- Runt Trigger menu
- Timeout Trigger menu

Edge Trigger configuration menu

Use the edge **Trigger** menu to trigger the instrument when a signal rises and/or falls through a specified level.

To open the Edge trigger menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Edge**.

Settings panel (Edge Trigger configuration menu) fields and controls

Field or control	Description
Trigger Type	Set to **Edge**.

Field or control	Description
Source	Lists the source channel. **AC Line** source (**Edge** trigger): This provides a trigger source that is aligned with the line frequency of the input. This can cause the oscilloscope to trigger close to 60 times per second. 50-60 Hz line frequencies are supported.
Coupling	Set the conditioning to apply to the source signal trigger circuit from the source signal. **DC** coupling passes all input signals directly to the trigger circuitry. **HF Reject** coupling attenuates signals above 50 kHz before passing the signal to the trigger circuitry. **LF Reject** coupling attenuates signals below 50 kHz before passing the signal to the trigger circuitry. **Noise Reject** coupling provides stable triggering by increasing the trigger hysteresis. Increased hysteresis reduces the trigger sensitivity to noise so it may require greater signal amplitude.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Slope	Sets the signal transition direction to detect (rising or falling).

Other trigger types

- Bus Trigger Menu
- Pulse Width Trigger menu
- Rise Fall Time Trigger menu
- Runt Trigger menu
- Timeout Trigger menu

Pulse Width Trigger configuration menu

Use the Pulse Width Trigger to trigger on specific pulse width conditions, including when a pulse width is within or outside a range of specified times. To open the Pulse Width trigger configuration menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Pulse Width**.

Settings panel (Pulse Width Trigger configuration menu) fields and controls

Field or control	Description
Source	Lists the source channel.
Trigger When	• **< Limit**: A pulse width is less than the specified time limit. • **> Limit**: A pulse width is greater than the specified time limit. **Inside Range**: A pulse width is in the specified time range.

Field or control	Description
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Time Limit	Sets the time period condition to be met.
High Time Limit (Trigger When = Inside Range)	Sets the longest acceptable pulse width time period for the range condition.
Low Time Limit (Trigger When = Inside Range)	Sets the shortest acceptable pulse width time period for the range condition.
Polarity	Sets the polarity of the pulse to detect positive and negative pulse only.

Other trigger types

- Bus Trigger Menu
- Edge Trigger menu
- Rise Fall Time Trigger menu
- Runt Trigger menu
- Timeout Trigger menu

Timeout Trigger configuration menu

Use the Timeout Trigger to trigger on a waveform when an expected signal does not transition within a specified period of time, such as when a signal gets stuck either high or low.

To open the Timeout trigger menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Timeout**.

Settings panel (Timeout Trigger configuration menu) fields and controls

Field or control	Description
Source	Lists the source channel.
Trigger When	• **Stays High**: The signal stays above the specified threshold level longer than the specified time. • **Stays Low**: The signal stays below the specified threshold level longer than the specified time. • **Either**: The signal stays above or below the specified threshold level longer than the specified time.
Threshold	Sets the amplitude level that the signal must pass through to be considered a valid transition.

Field or control	Description
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Time Limit	Sets the time period condition to be met.

Other trigger types

- Bus Trigger Menu
- Edge Trigger menu
- Pulse Width Trigger menu
- Rise Fall Time Trigger menu
- Runt Trigger menu

Runt Trigger configuration menu

Use the Runt trigger to trigger on waveforms where a short pulse crosses one threshold but fails to cross a second threshold before recrossing the first.

To open the Runt trigger configuration menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Runt**.

Settings panel (Runt Trigger configuration menu) fields and controls

Field or control	Description
Source	Lists the source channel.
Trigger When	• **Occurs**: A runt signal event occurs. • **< Limit**: A runt signal event occurs that has a pulse width less than the specified time limit. • **> Limit**: A runt signal event occurs that has a pulse width greater than the specified time limit.
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Time Limit	Sets the time period condition to be met.
Upper Threshold	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.

Other trigger types

- Bus Trigger Menu
- Edge Trigger menu
- Pulse Width Trigger menu
- Rise Fall Time Trigger menu
- Timeout Trigger menu

Rise/Fall Time Trigger configuration menu

Use the Rise/Fall Time trigger to trigger when the rise or fall time of a signal is less than, greater than, equal to, or not equal to a specified time limit.

To open the Rise/Fall Time trigger configuration menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Rise/Fall Time**.

Settings panel (Rise/Fall Time Trigger configuration menu) fields and controls

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search.
Trigger When	• **< Limit**: A signal has a rise/fall time less than the specified time limit. • **> Limit**: A signal has a rise/fall time greater than the specified time limit.
In Range	When In Range is selected options are: ▪ High time limit sets the longest acceptable pulse width time period for the range condition. ▪ Low time limit sets the shortest acceptable pulse width time period for the range condition.
Time Limit	Sets the time period condition to be met.
Slope	Sets the signal transition direction to detect (rising or falling).
Upper Threshold	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.

Other trigger types

- Bus Trigger Menu
- Edge Trigger menu
- Pulse Width Trigger menu
- Runt Trigger menu
- Timeout Trigger menu

Bus Trigger configuration

Use the Bus trigger menus to trigger on bus-related events (Start, Stop, Missing Ack, Address, Data, and so on).

NOTE: You must add a bus to the Waveform view before you can trigger on it. Add a math, reference, or bus waveform.

To open the Bus trigger menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Bus**.
3. Select the bus on which to trigger in the **Source** field.

Use the following links to access trigger setting information for specific serial buses.

I2C serial bus trigger settings panel

RS-232 serial bus trigger settings panel

SPI serial bus trigger settings panel

Other trigger types

- Edge Trigger menu
- Pulse Width Trigger menu
- Rise/Fall Time Trigger menu
- Runt Trigger menu
- Timeout Trigger menu

I2C serial bus trigger settings panel

Field or control	Description
Source	Select the **I2C** bus on which to trigger.
Trigger On	Select the type of information on which to trigger.
Direction	Sets the transfer direction on which to trigger (Read, Write). Available when **Trigger On** is **Address**.
Addressing Mode	Sets the slave device address length (7 bits or 10 bits long). Available when **Trigger On** is **Address** or **Address & Data**.
Address	Sets the address pattern on which to trigger. The number of bits shown depends on the **Address Mode** setting. Tap the **Binary** or **Hex** field and use the **A** knob to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Trigger On** is **Address** or **Address & Data**.
Data Bytes	Sets the number of data bytes on which to trigger (1 to 5 bytes). Use the A knob to change the value. Available when **Trigger On** is **Data** or **Address & Data**.

Field or control	Description
Data	Sets the data pattern on which to trigger. The number of bits shown depends on the **Data Bytes** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Trigger On** is **Data** or **Address & Data**.
A , B knobs controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s). Or double-tap on the field and use the virtual keypad to enter values.

RS-232 serial bus trigger settings panel

Field or control	Description
Source	Select the **RS232** bus on which to trigger.
Trigger On	Select the type of information on which to trigger.
Data Bits	Sets the number of data bytes (1 byte = 8 bits) on which to trigger (one to ten bytes). Use the **A** knob to change the value. Available when **Trigger On** = **Data**.
Data	Sets the data on which to trigger. The number of bits shown depends on the **Data Bits** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Trigger On** = **Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).

SPI serial bus trigger settings panel

Field or control	Description
Source	Select the **SPI** bus on which you want to trigger.
Trigger On	Select the type of information on which to trigger.
Word Size	Sets the number of bits in a word on which to trigger (4 to 32). Use the **A** knob to change the value. Available when **Trigger On** = **Data**.
FrameLen	Sets the number of data words on which to trigger (1 to 32). Use the **A** knob to change the value.

Field or control	Description
MOSI Data	Sets the data pattern on which to trigger. The number of bits shown depends on the **Word Size** and **FrameLen** settings. Double-tap on the field, .tap the **Binary** or **Hex** field and use the virtual keypad to enter values. Available when **Trigger On** = **Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).

Virtual Keyboard

Use the onscreen virtual keyboard to enter textual information such as a file path, file name, label text, or on-screen callout.

To access the virtual keyboard, double-tap in a menu or dialog text input box. Enter your text and tap **Enter** to close the keyboard and add your text to the menu or dialog field.

Tap **ESC**, **Cancel**, or anywhere outside the keyboard, to dismiss it without adding text to the input box.

Single-tap in the keyboard text field to position the insertion cursor at that location.

Touch and drag the title bar to move the keyboard on the screen.

Other than the standard English language, the virtual keyboard also supports German, French, and Italian languages for interacting with the oscilloscope.

Bus trigger radix-specific virtual keypads

Use the virtual Logic keypad to edit bus logic values for trigger settings. Using a logic keypad is faster to set larger logic trigger values than using the multipurpose knobs in the trigger menu.

To open the virtual Logic keypad, double-tap inside a bus trigger field that requires logic values. Which field you select (Binary, Hex, and so on) sets which Logic keypad is displayed.

Virtual Keypad fields and controls

Field or control	Description
Def	Sets all digits to 0.
<	Moves the insertion point left and highlights the character that can be edited.
>	Moves the insertion point right and highlights the character that can be edited.
∧	Moves the insertion point up and highlights the character that can be edited.
∨	Moves the insertion point down and highlights the character that can be edited.
Radix keypad	Provides a keypad associated with the logic entry radix field that was double-tapped in the Trigger menu. Tapping a key sets the selected digit to the specified value and moves the selected digit indicator to the next (right) digit.

Field or control	Description
OK	Closes the number pad and assigns the entered value to the field. Also updates all other formats in the trigger menu so that all formats are indicating the same value.

Virtual Keypad

Use the virtual Keypad to enter numeric values and units for settings.

To open the virtual keypad, double-tap inside a field that requires numeric values.

Virtual Keypad fields and controls

Field or control	Description
Cancel	Closes the logic keypad without saving entered values.
Clear	Clears all values from the input entry field.
Exp	Lets you enter exponential notation entries.
Max	Enters the maximum value allowed for this setting.
Min	Enters the minimum value allowed for this setting.
Bksp ←	Deletes characters to the left of the insert text marker position.
Enter	Closes the number pad and assigns the entered value to the field.
±	Tap the button to set a numeric value to a positive (default) or negative value.
Unit buttons	Use to set the units of the entered value.

Waveform View configuration menu

Use this menu to configure the Waveform View, waveform interpolation method, persistence, and other parameters.

To open the Waveform View menu, double-tap anywhere in the Waveform View screen.

Waveform View menu fields and controls

Fields or controls	Description
Display Mode	Sets how waveforms are shown on the screen. **Overlay** mode displays all waveforms on the screen, overlaid on each other. This is the traditional way that oscilloscopes displayed waveforms, and lets you overlay waveforms to do direct comparisons of waveform shape. The grid vertical position and scale values change for each selected waveform.

Fields or controls	Description
Interpolation	Selects the method used to display record points between sampled points. **Sin(x)/x** computes record points along a curve between the actual acquired samples. This form of interpolation is useful when acquiring rounded waveforms such as sine waves. It is good for general-purpose uses but may introduce overshoot or undershoot in signals with fast rise times. This interpolation is also useful for looking at high-frequency signals, especially where the frequency components are just below the Nyquist frequency. **Linear** computes record points between actual acquired samples using a straight-line fit. This interpolation is useful for measuring waveforms with fast rise times, such as pulse trains.
Persistence	Sets the length of time data points are displayed on screen before being erased. **Off** sets the record points to appear for the current acquisition only. **Infinite** continuously accumulates record points on the waveform until you change one of the acquisition display settings or clear the acquisition memory. Use infinite persistence for displaying record points that may occur outside the normal acquisition envelope. **Variable** lets you specify a time length to retain data points on screen. Each record point decays independently according to the time interval. **Auto** sets the **Waveform Intensity** field to control the persistence time.
Variable Persistence Time (Persistence = Variable)	Sets the length of time data points remain displayed. Tap the field and use the A knob to adjust, or double-tap and use the keypad to set a time.
Waveform Style	Sets how waveforms are drawn on the screen. **Vectors** draws waveforms with lines between record points. **Dots** draws waveform record points as dots on the screen, and adds crosshair markers to real sampled points.
Waveform Intensity	Sets the brightness of the waveform. Tap the field and use the A knob to set the waveform intensity.
Graticule Style	Sets the type of graticule to display. **Grid** shows a traditional grid on the instrument display. This is the default grid mode. **Full** displays the same style graticule lines for both horizontal and vertical scale units. **None** turns off the graticule, including the vertical scale readouts.
Graticule Intensity	Sets the brightness of the graticule. Tap the field and use the A knob to set the graticule intensity.

Waveform acquisition

Acquisition concepts

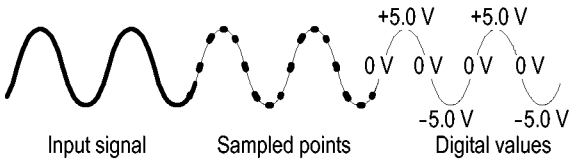
The Acquisition system sets which data points are used to acquire waveforms.

Acquisition hardware

Before a signal is displayed, it must pass through the input channel where it is scaled and digitized. Each channel has a dedicated input amplifier and digitizer. Each channel produces a stream of digital data from which the instrument extracts waveform records.

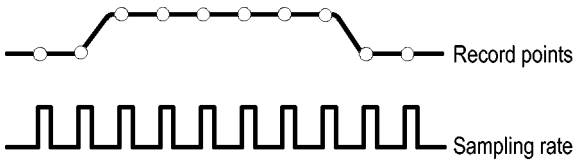
Sampling process

Acquisition is the process of sampling an analog signal, converting it into digital data, and assembling it into a waveform record, which is then stored in acquisition memory.



Real-Time sampling

In real-time sampling, the instrument digitizes all of the points it acquires using one trigger event. Use real-time sampling to capture single-shot or transient events.



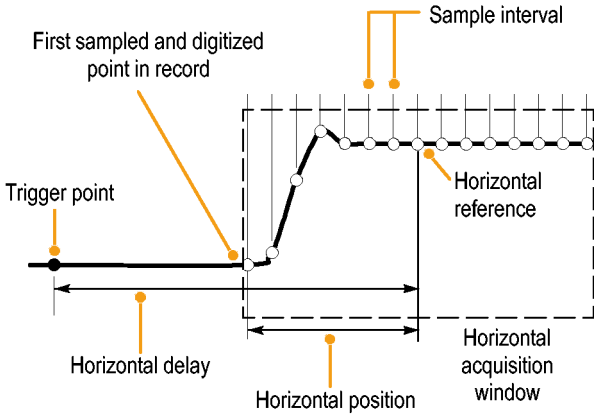
Interpolated Real-Time sampling

In interpolated real-time sampling, the instrument digitizes all of the points it acquires using one trigger event. If the instrument cannot acquire enough samples for a complete waveform at the maximum real-time sample rate, it interpolates. Use interpolated real-time sampling to capture single-shot or transient events.

Waveform record

The instrument builds the waveform record through use of the following parameters:

- Sample interval: The time between sample points.
- Record length: The number of samples required to fill a waveform record.
- Trigger point: The zero time reference in a waveform record.
- Horizontal position: When horizontal delay is off, the horizontal position is a percentage of the waveform record between 0 and 99.9 percent. The trigger point and the horizontal reference are at the same time in the waveform record. For example, if the horizontal position is 50 percent, then the trigger point is in the middle of the waveform record. When horizontal delay is on, the time from the trigger point to the horizontal reference is the horizontal delay.



Interpolation

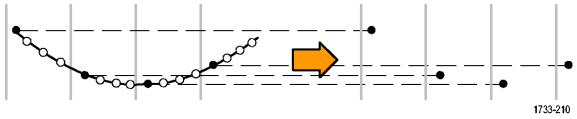
Your instrument can interpolate between the samples it acquires when it does not have all of the actual samples it needs to fill the waveform record. Linear interpolation computes record points between actual acquired samples by using a straight line fit.

Sin(x)/x interpolation computes record points using a curve fit between the actual values acquired. Sin(x)/x interpolation is the default interpolation mode because it requires fewer actual sample points than linear interpolation to accurately represent the waveform.

Acquisition modes

Acquisition is the process of sampling an analog signal, converting it into digital data, and assembling it into a waveform record, which is then stored in acquisition memory. The acquisition mode determines how the waveform record points are calculated from the sampled waveform data.

How the acquisition modes work:

Acquisition mode	
Sample mode retains the first sampled point from each acquisition interval. Sample is the default mode. The instrument does no post processing of the acquired samples in this mode.	
Average mode calculates the average value for each record point over many acquisitions. Average uses Sample mode for each individual acquisition. Use Average mode to reduce random noise.	
Roll mode starts at 40 ms/div. To disable Roll Mode, set **Trigger Mode** to **Normal**.	

Waveform sample interpolation

When the sample density falls to less than one sample per display column, the instrument must calculate intermediate points to display a waveform. This process is called interpolation.

There are two options for interpolation:

- Sin(x)/x interpolation. Computes record points using a curve fit between the actual values acquired. It assumes that all the interpolated points fall along that curve.
- Linear interpolation. Computes record points between actual acquired samples by using a straight line fit. It assumes that all the interpolated points fall in their appropriate point in time on that straight line.

Coupling

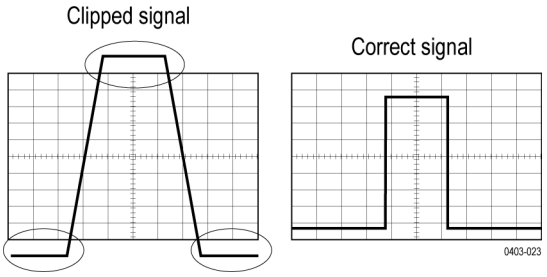
Coupling determines whether an input signal is directly connected to the input channel (DC coupling), connected through a DC blocking capacitor (AC coupling), or not connected at all (GND coupling).

All probes expect a specific coupling and input termination. Both coupling and input termination are displayed on the screen.

Scaling and positioning

The scaling and positioning controls determine the portion of the input signal received by the acquisition system. Set the vertical scaling, positioning, and DC offsets to display the features of interest on your waveform and to avoid clipping.

Each waveform Slice or Graticule contains ten major divisions. This represents the maximum digitizing range of the instrument for any given vertical scale. Vertical waveform data that is outside (above and/or below) of the waveform Slice or Graticule is clipped; that is, the data values exceed the digitizing capability of the ADC at the current settings. This causes inaccuracies in amplitude-related measurements. For more information see Vertical acquisition considerations.



Set the horizontal scale, position, and resolution (record length) to include the acquired waveform record waveform attributes of interest with good sampling density on the waveform. These settings define the horizontal acquisition window, described in Horizontal acquisition considerations.

NOTE: The terms vertical acquisition window and horizontal acquisition window refer to the vertical and horizontal range of the segment of the input signal that the acquisition system acquires. The terms do not refer to any display windows on screen.

Vertical acquisition considerations

You can set the vertical scale, position, and offset of each channel independently of other channels. Vertical scale and offset specify the vertical parameters of the waveform display for each channel. The oscilloscope only acquires signals that fall within these parameters.

You can set the vertical scale, position, and offset of each channel independently of other channels. Vertical scale and offset specify the vertical acquisition window for each channel. Parts of the signal amplitude that fall within the vertical window are acquired; parts outside (if any) are not.

The offset control subtracts a constant DC level from the input signal before the vertical scale factor is applied, and the vertical position control adds a constant number of divisions of signal after the scale factor is applied to the resulting difference.

The vertical scale and position controls have the following effects on the waveform display and the displayed waveform:

The vertical volts per division you set determines the vertical size of the waveform display, allowing you to scale it to contain all of a waveform amplitude or only part.

NOTE: Amplitude-related automatic measurements (for example, peak-to-peak and RMS) will be accurate for vertical windows if the waveform is not clipped (that is, the waveforms are acquired). But if signal amplitude were to extend outside the vertical acquisition window, the data acquired is clipped. Clipped data causes inaccurate results if used in amplitude-related automatic measurements. Clipping also causes inaccurate amplitude values in waveforms that are stored or exported for use in other programs.

If the scale of a math waveform is changed so that the math waveform is clipped, it will affect the amplitude measurements on that math waveform as follows:

- The vertical position adjusts the display of the graticule relative to the waveform display (position is a display control). That is all position does; it does not determine what data is acquired as do vertical scale and offset.
- As you vary vertical offset, the middle voltage level moves relative to zero. This moves the vertical acquisition window up and down on the waveform. With input signals that are smaller than the window, it appears the waveform moves in the waveform view. Actually, the offset moves the middle of the vertical acquisition window up and down on the input signal. Offset moves the acquisition window to control the portion of the waveform amplitude the window captures.
- Applying a negative offset moves the vertical range down relative to the DC level of the input signal. Likewise, applying a positive offset moves the vertical range up.

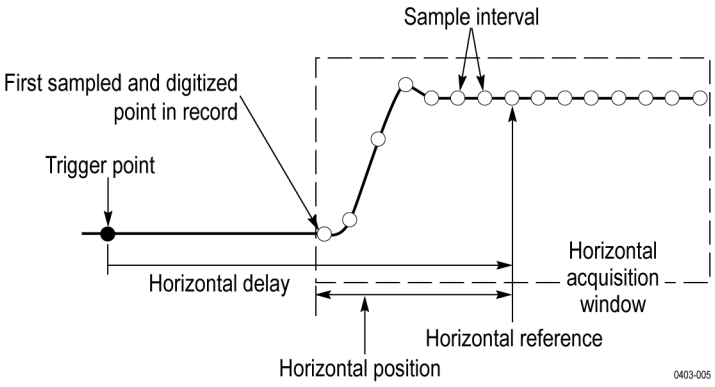
Horizontal acquisition considerations

The instrument lets you define the horizontal waveform display parameters that determine which segment of an incoming signal becomes the waveform record, following acquisition. For additional information, see "Waveform record" in Acquisition concepts (on page 118).

These common parameters specify a horizontal scale and position that is applied to all channels simultaneously.

These parameters are shown in the next figure (horizontal window with delay on):

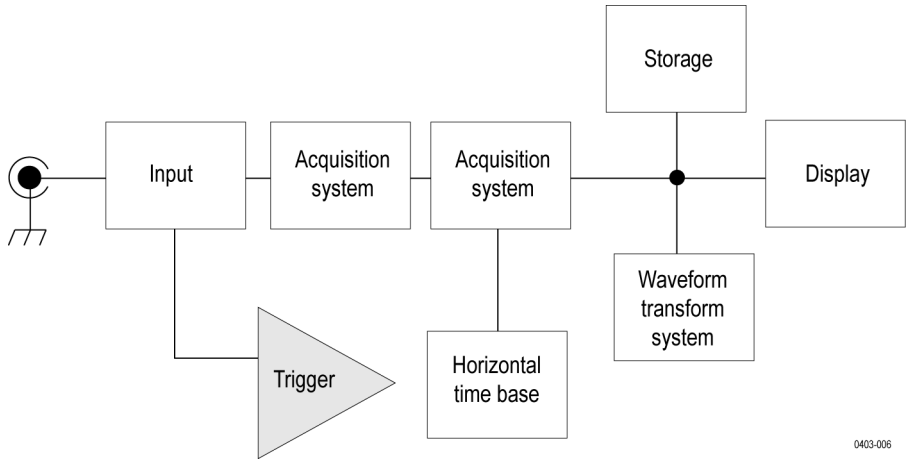
- The trigger position determines where the trigger event will be located in the waveform record. To see more pretrigger data move your trigger position to the right on the graticule.
- The horizontal position determines the number of pretrigger and posttrigger samples. Samples before the trigger point are pretrigger samples and those after the trigger point are posttrigger samples. When Delay is off, the horizontal position is the same as the trigger position.
- The horizontal delay determines the time from the trigger point to the Horizontal Reference.
- The horizontal scale determines the horizontal size of the display relative to any waveform, allowing you to scale it to contain a waveform edge, a cycle, or several cycles.



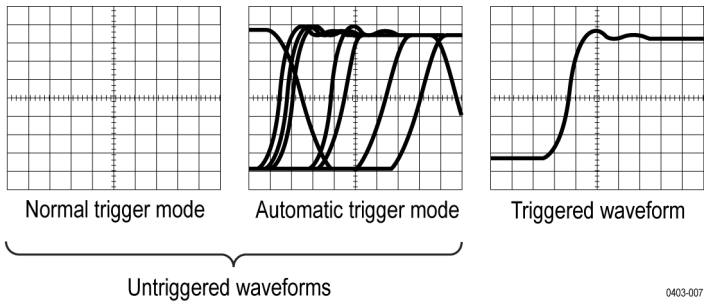
Triggering concepts

User selected trigger conditions are used to capture waveforms for measurement and analysis.

The next figure shows how triggers fit into the overall instrument operation.



Trigger helps you to capture meaningful waveforms to display on screen. This instrument has simple edge triggers as well as a variety of advanced triggers.



The trigger event

The trigger event establishes the time-zero point in the waveform record. All waveform record data are located in time with respect to that point. The instrument continuously acquires and retains enough sample points to fill the pretrigger portion of the waveform record (that part of the waveform that is displayed before or to the left of the triggering event on screen).

When a trigger event occurs, the instrument starts acquiring samples to build the post-trigger portion of the waveform record (displayed after or to the right of the trigger event). Once a trigger is recognized, the instrument will not accept another trigger until the acquisition is complete and the holdoff time has expired.

Trigger sources

The trigger source provides the signal that triggers acquisition. Use a trigger source that is synchronized with the signal that you are acquiring and displaying. You can derive your trigger from the following sources:

- **Input channels**: Analog input channels are the most commonly used trigger sources. You can select any of the input channels. The channel that you select as a trigger source will function whether it is displayed or not.
- **Bus**: This source is used to trigger a parallel bus or a serial bus. You can include any combination of analog channels to build a parallel bus, or use any channel as a component in a serial bus.

Trigger types

The available trigger types include:

Bus: This trigger is used to set up parallel and serial buses. A bus trigger event occurs when the instrument detects a bus pattern that you set in a bus trigger configuration menu. You must add a bus to the Waveform view before you can set the trigger parameters for that bus.

Edge: This is the simplest and most commonly used trigger type, used with analog signal. An edge trigger event occurs when the trigger source passes through a specified voltage level in the specified direction (rising or falling signal voltage). When the edge trigger source is set to **AC Line**, the instrument uses the line frequency of the supply power to generate a trigger.

Pulse Width: Trigger on pulses that are inside or outside a specified time range. Can trigger on positive or negative pulses.

Rise/Fall Time: Trigger on pulse edges that traverse between two thresholds at faster or slower rates than the specified time. The pulse edges can be positive or negative.

Runt: Use Runt trigger to trigger on a pulse amplitude that crosses one threshold but fails to cross a second threshold before recrossing the first. It can detect any positive or negative runts, or those that are inside or outside a specified time range. These pulses can also be qualified by the logical state of other channels.

Timeout: Trigger when no edge transition is detected within a specified time.

Trigger modes

The trigger mode determines how the instrument behaves in the absence of a trigger event:

- **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the instrument does not acquire a waveform, and the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
- **Auto** trigger mode enables the instrument to acquire a waveform even if a trigger does not occur. Auto mode uses a timer that starts after a trigger event occurs. If another trigger event is not detected before the time out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting.

Auto mode, when forcing triggers in the absence of valid triggering events, does not synchronize the waveform on the display. In other words, successive acquisitions are not triggered at the same point on the waveform. If valid triggers occur, the display becomes stable.

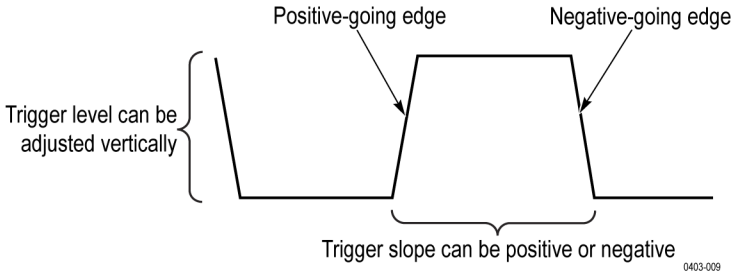
Trigger coupling

Trigger coupling determines what part of the signal is passed to the trigger circuit. Edge triggering can use all available coupling types: DC, Low Frequency Rejection, High Frequency Rejection, and Noise Rejection. All of the advanced trigger types use DC coupling only.

- **DC**. This coupling passes all input signals to the trigger circuitry.
- **HF Reject**. This coupling attenuates signals above 50 kHz before passing the signal to the trigger circuitry.
- **LF Reject**. This coupling attenuates signals below 50 kHz before passing the signal to the trigger circuitry.
- **Noise Reject**. This coupling provides stable triggering by increasing the trigger hysteresis. Increased hysteresis reduces the trigger sensitivity to noise but may require greater signal amplitude.

Trigger slope and level

The slope control determines whether the instrument finds the trigger point on the rising or the falling edge of a signal. The level control determines where on that edge the trigger point occurs. See the next figure.



Trigger position in waveform record

Trigger position is an adjustable feature that defines where the trigger occurs on the waveform record. It lets you choose how much the instrument acquires before and after the trigger event. The part of the record that occurs before the trigger is the pretrigger portion. The part that occurs after the trigger is the posttrigger portion. A longer posttrigger period may be useful when you want to see the effects an event has on your system under test.

Pretrigger data can be valuable when troubleshooting. For example, if you are trying to find the cause of an unwanted glitch in your test circuit, you can trigger on the glitch and make the pretrigger period large enough to capture data before the glitch. By analyzing what happens before the glitch, you may uncover information that helps you find the source of the glitch.

Bus triggering concepts

A bus trigger occurs when a supported instrument detects a bus pattern that you specify for a bus cycle you select for a serial bus.

You can set the instrument to trigger on an SPI bus when the instrument detects an SS Active bus cycle or Data.

You can set the instrument to trigger on an I2C bus when the instrument detects a Start, Stop, Repeated Start, Missing Ack, Address, Data, or Address + Data bus cycle or activity.

For all the serial standard buses, you can also set the component threshold levels through the Bus Setup menu.

Pulse width trigger concepts

A pulse width trigger occurs when the instrument detects a pulse that is inside or outside some specified time range. The instrument can trigger on positive or negative width pulses.

Timeout trigger concepts

A timeout trigger occurs when the instrument does not detect an expected pulse transition within a user specified period of time, such as when a signal gets stuck either high or low. If the pulse transition occurs prior to a specified timeout time (the expected case), then no trigger results.

Runt trigger concepts

A runt trigger occurs when the instrument detects a short pulse that crosses one threshold but fails to cross a second threshold before recrossing the first. You can set the instrument to detect any positive or negative runts, or those that are inside or outside a specified time range.

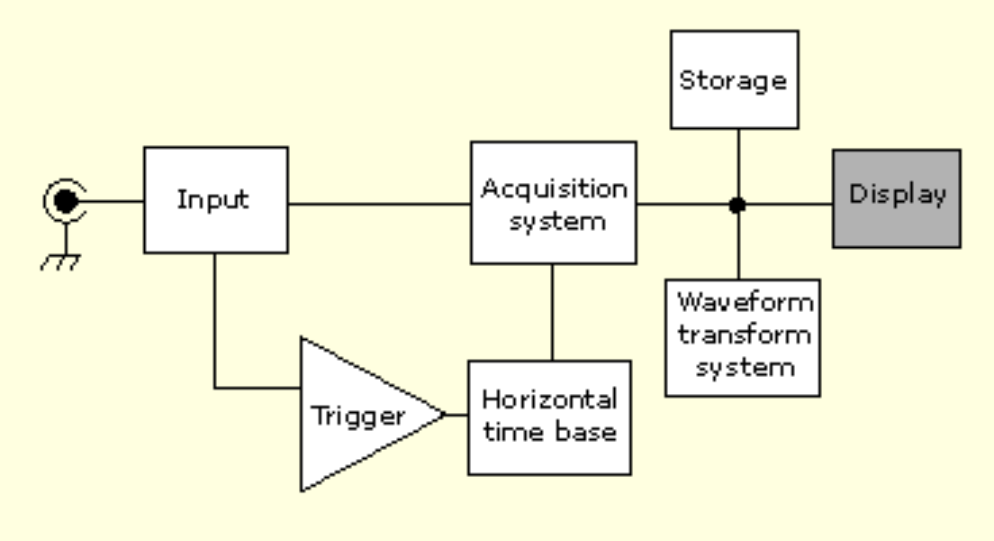
Rise/Fall time trigger concepts

Rise/Fall time triggering is based on the slope (change in voltage/change in time) of a pulse edge.

Use the Rise/Fall trigger to trigger the instrument on pulse edges that traverse between two thresholds at faster or slower rates than the specified time. You can set up the instrument to trigger on positive or negative edges.

Waveform display concepts

This instrument includes a flexible, customizable display that lets you control how waveforms appear. The figure shows how the display features fit into the overall instrument operation.



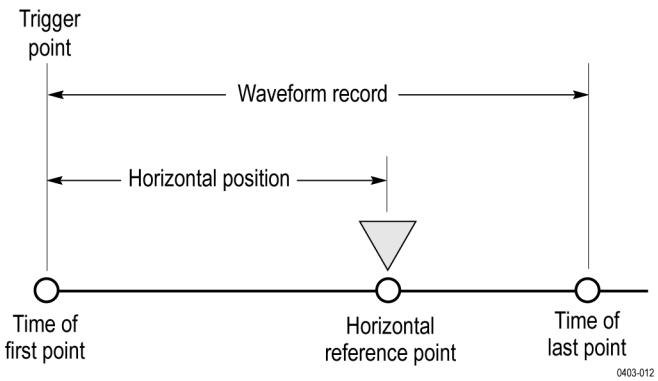
The display shows analog, math, reference, and bus waveforms. The waveforms include channel markers, individual waveform graticule readings, and trigger source and level indicators. You can overlay all the waveforms on the screen (traditional waveform view).

Waveform preview mode

The waveform preview attempts to show what the next acquisition will look like when the acquisition is delayed due to slow triggers or long acquisition duration, or when the acquisitions have stopped. Waveform preview recalculates math waveforms, but does not represent changes in trigger levels, trigger modes, or different acquisition modes.

Horizontal position and the horizontal reference point

The time value you set for horizontal position is measured from the trigger point to the horizontal reference point. This is not the same as the time value from the trigger point to the start of the waveform record, unless you set the horizontal reference to 0%. See the next figure.



Learn more by viewing these topics:

Step by step procedure to display waveforms. (on page 36)