Tektronix®

5 Series MSO
MSO54, MSO56, MSO58, MSO58LP
Printable Help

Supports 5 Series MSO Product Firmware V1.4.x and above.

077-1303-01

Contacting Tektronix

Tektronix, Inc.
14150 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.

Table of Contents

Product documents and support

Accessories

Options

Install your instrument

Getting acquainted with your instrument

Configure the instrument

Analog channel operating basics

Acquiring digital signals

Advanced triggering

Setting waveform display parameters

Zooming on waveforms

Customizing measurements

Saving and recalling information

Menus and dialog boxes

Waveform acquisition concepts

Trigger concepts

Waveform display concepts

Measurement concepts

Measurement algorithms

References

Welcome to the 5 Series MSO Help

The 5 Series MSO oscilloscopes (MSO54, MSO56, MSO58, and MSO58LP) are 4-, 6-, and 8-channel oscilloscopes with FlexChannel™ technology, enabling you to efficiently and cost-effectively perform mixed signal debugging on virtually any design.

MSO54, MSO56, MSO58 Key features and benefits

- Bandwidths from 350 MHz to 2 GHz
- 4-, 6-, and 8-channel models with FlexChannel™ inputs
- Each FlexChannel input is dual-purpose, letting you connect either an analog probe (TekVPI® or BNC) or an eight-channel digital probe (the TLP058 FlexChannel Logic Probe)
- Large 15.6” HD (1920 x 1080 pixel) capacitive touch-screen display
- User interface designed to optimize touch screen use
- Maximum 6.25 GS/s sample rate
- 62.5 M points record length on all channels (optional 125 M record length available)
- 500,000 waveforms/second maximum waveform capture rate
- FastFrame™ segmented memory acquisition uses multiple trigger events to capture widely spaced events of interest at high sample rates while conserving acquisition memory
- No set limit on the number of math, reference, and bus waveforms you can display
- Integrated options include 50 MHz arbitrary/function generator (AFG), Digital Voltmeter (DVM), and a trigger frequency counter
- Optional advanced serial bus triggering and analysis functions let you trigger on and decode I^2C, SPI, USB 2.0, CAN, CAN FD, LIN, FlexRay, RS-232/422/485/UART, I^2S, Left Justified (LJ), Right Justified (RJ), TDM, Ethernet, MIL-STD-1553, and ARINC429 buses
- Option 5-PWR provides power analysis measurements and plots
- Option 5-CMAUTOEN adds Automotive Ethernet (100BASE-T1) compliance testing.

MSO58LP Key features and benefits

- 1 GHz bandwidth
- 8-channels with FlexChannel™ inputs
- Each FlexChannel input is dual-purpose, letting you connect either an analog probe (TekVPI® or BNC) or an eight-channel digital probe (the TLP058 FlexChannel Logic Probe)
- Low profile instrument with no display, designed for 2U rack mount space and remote operation
- Maximum 6.25 GS/s sample rate
- 125 M points record length on all channels
- 500,000 waveforms/second maximum waveform capture rate

Actual number of waveforms depends on available system memory.
Option 5-CMAUTOEN requires option 5-WIN (SSD with Microsoft Windows 10 operating system)

- FastFrame™ segmented memory acquisition uses multiple trigger events to capture widely spaced events of interest at high sample rates while conserving acquisition memory
- No set limit on the number of math, reference, and bus waveforms you can acquire [1]
- Integrated options include 50 MHz arbitrary/function generator (AFG), Digital Voltmeter (DVM), and a trigger frequency counter
- Optional advanced serial bus triggering and analysis functions let you trigger on and decode I^2C, SPI, USB 2.0, CAN, CAN FD, LIN, FlexRay, RS-232/422/485/UART, I^2S, Left Justified (LJ), Right Justified (RJ), TDM, Ethernet, MIL-STD-1553, and ARINC429 buses
- Option 5-PWR provides power analysis measurements and plots

5 Series MSO Help Version 20171018-16:00 for FW version 1.4.x

Product documents and support

Related documents

Use the related documents to access information on how to remotely program or operate the instrument, understand theory of operation, replace suspected modules, and do other tasks.

MSO54, MSO56, MSO58 documents

To learn about	Use this document
How to use instrument functions	*5 Series MSO (MSO54, MSO56, MSO58, MSO58LP) Help* (Tektronix part number 077-1303-xx; Printable version of the instrument Help; available at www.tektronix.com/downloads) *5 Series MSO (MSO54, MSO56, MSO58) Installation and Safety Manual* (Tektronix part number 071-3514-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. Russian language version available to download from the Tektronix web site (Tektronix part number 077-1361-xx)
How to remotely control the instrument	*5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx; available at www.tektronix.com/downloads)
Instrument specifications and procedures to verify the instrument meets specifications	*5 Series MSO MSO54, MSO56, MSO58, MSO58LP Specifications and Performance Verification Technical Reference* (Tektronix part number 077-1306-xx; available at www.tektronix.com/downloads)
Instrument theory of operation, troubleshooting, disassembly, and replaceable parts	*5 Series MSO MSO54, MSO56, MSO58 Service Manual* (Tektronix part number 077-1307-xx; available at www.tektronix.com/downloads)
Installing the instrument in a rack	*RM5 Rack Mount Kit Instructions* (Tektronix part number 071-3523-xx; available at www.tektronix.com/downloads)
Using the TLP058 Logic Probe	*TLP058 FlexChannel Logic Probe Instructions* (Tektronix part number 071-3515-xx; available at www.tektronix.com/downloads)

To learn about	Use this document
How to use instrument functions	*5 Series MSO MSO54, MSO56, MSO58 , MSO58LP Help* (Tektronix part number 077-1303-xx; Printable version of the instrument Help; available at www.tektronix.com/downloads) *5 Series MSO MSO58LP Installation and Safety Manual* (Tektronix part number 071-3568-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. Russian language version available to download from the Tektronix web site (Tektronix part number 077-1404-xx)
How to remotely control the instrument	*5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx; available at www.tektronix.com/downloads)
Instrument specifications and procedures to verify the instrument meets specifications	*5 Series MSO MSO54, MSO56, MSO58 , MSO58LP Specifications and Performance Verification Technical Reference* (Tektronix part number 077-1306-xx; available at www.tektronix.com/downloads)
Instrument theory of operation, troubleshooting, disassembly, and replaceable parts	*5 Series MSO MSO58LP Service Manual* (Tektronix part number 077-1405-xx; available at www.tektronix.com/downloads)
Using the TLP058 Logic Probe	*TLP058 FlexChannel Logic Probe Instructions* (Tektronix part number 071-3515-xx; available at www.tektronix.com/downloads)

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 the Web site. See Contacting Tektronix on page 0 for more information or assistance with your product.

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, 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

Accessories

Standard accessories

MSO54, MSO56, MSO58 standard accessories

Item	Quantity	Tektronix part number
5 Series MSO (MSO54, MSO56, MSO58) Installation and Safety Manual	1	071-3514-xx
TPP0500B Passive Voltage Probe (500 MHz bandwidth). Shipped with 350 MHz and 500 MHz models.	One per channel	TPP0500B
TPP1000 Passive Voltage Probe (1 GHz bandwidth). Shipped with 1 GHz and 2 GHz models.	One per channel	TPP1000
Front cover	1	200-5406-xx
Accessory pouch (attached to front cover)	1	016-2106-xx
Mouse (wired with USB connector)	1	119-7054-xx
Power cord	1	Depends on region
Calibration certificate	1	N/A
Report of factory installed licenses	1	N/A

MSO58LP standard accessories

Item	Quantity	Tektronix part number
5 Series MSO MSO58LP Installation and Safety Manual	1	071-3568-xx
Power cord	1	Depends on region
Calibration certificate	1	N/A
Report of factory installed licenses	1	N/A

Recommended accessories

See the Tektronix Web site (www.tek.com) for the latest information on recommended accessories for this product.

MSO54, MSO56, MSO58 recommended accessories

Accessory	Tektronix part number
Hard transit case	HC5
Rackmount kit	RM5
Mini keyboard	119-7275-xx
TekVPI-to-TekProbe BNC adapter	TPA-BNC
Deskew fixture	TEK-DPG
5 Series MSO (MSO54, MSO56, MSO58) Service Manual as a PDF file, download from the Tektronix Web site	077-1307-xx
5 Series MSO (MSO54, MSO56, MSO58, MSO58LP) Programmer Interface Manual as a PDF file, download from the Tektronix Web site	077-1305-xx
5 Series MSO (MSO54, MSO56, MSO58, MSO58LP) Specifications and Performance Verification Technical Reference as a PDF file, download from the Tektronix Web site	077-1306-xx

MSO58LP recommended accessories

Accessory	Tektronix part number
Mini keyboard	119-7275-xx
TekVPI-to-TekProbe BNC adapter	TPA-BNC
Deskew fixture	TEK-DPG
5 Series MSO MSO58LP Benchtop Conversion kit (Tektronix part number). The kit includes chassis feet and a handle, and lets you stack instruments on a bench	020-3180-xx
5 Series MSO MSO58LP Service Manual as a PDF file, download from the Tektronix Web site	077-1405-xx
5 Series MSO (MSO54, MSO56, MSO58, MSO58LP) Programmer Interface Manual as a PDF file, download from the Tektronix Web site	077-1305-xx
5 Series MSO (MSO54, MSO56, MSO58, MSO58LP) Specifications and Performance Verification Technical Reference as a PDF file, download from the Tektronix Web site	077-1306-xx

Recommended probes

See the Tektronix Web site (www.tek.com) for the latest information on supported probes for this product.

Probes

Probe model	Description
TAP1500	1.5 GHz TekVPI Single-ended Active FET probe
TAP2500	2.5 GHz TekVPI Single-ended Active FET probe
TAP3500	3.5 GHz TekVPI Single-ended Active FET probe
TCP0020A	50 MHz TekVPI 20 Ampere AC/DC current probe
TCP0030A	120 MHz TekVPI 30 Ampere AC/DC current probe
TDP0500	500 MHz TekVPI differential voltage probe with ±42 V differential input voltage
TDP1000	1 GHz TekVPI differential voltage probe with ±42 V differential input voltage
TDP1500	1.5 GHz TekVPI differential voltage probe with ±8.5 V differential input voltage
TDP3500	3.5 GHz TekVPI differential voltage probe with ±2 V differential input voltage
TCP0150	DC to 20 MHz, TekVPI 150 A AC/DC current probe
THDP0100	±6 kV, 100 MHz TekVPI high-voltage differential probe
THDP0200	±1.5 kV, 200 MHz TekVPI high-voltage differential probe
TMDP0200	±750 V, 200 MHz TekVPI high-voltage differential probe
P5100A	2.5 kV, 500 MHz, 100X high-voltage passive probe
P6015A	20 kV, 75 MHz high-voltage passive probe
TIVM1	±50 V, 1GHZ IsoVu probe 3 m
TIVM1L	±50 V, 1GHZ IsoVu probe 10 m
TRCP0300	Rogowski current probe, 9 Hz to 30 MHz, 250 mA to 300 A peak
TRCP0600	Rogowski current probe, 12 Hz to 3 MHz, 500 mA to 600 A peak
TRCP3000	Rogowski current probe, 500 mA to 3000 A peak

Options

Bandwidth options

These options let you order the oscilloscope with a specified bandwidth, or upgrade a purchased oscilloscope to a higher bandwidth.

There are no bandwidth options for the 5 Series MSO MSO58LP Low Profile instrument.

Installed bandwidth options

These options are ordered when purchasing the oscilloscope. You must order one of these options when purchasing the oscilloscope.

Option name	Description
5-BW-350 (default)	350 MHz bandwidth
5-BW-500	Upgrade from 350 MHz to 500 MHz bandwidth
5-BW-1000	Upgrade from 350 MHz to 1 GHz bandwidth
5-BW-2000	Upgrade from 350 MHz to 2 GHz bandwidth

Bandwidth upgrade options

These options can be ordered for already-purchased oscilloscopes. Some upgrades require sending the oscilloscope to a service center to replace hardware and recalibrate the instrument.

<table>
<tr><th>Option name</th><th>Description</th><th>Notes</th></tr>
<tr><td>SUP5-BW3T54</td><td>Bandwidth upgrade from 350 MHz to 500 MHz on 4-channel models</td><td rowspan="3">A license file to upgrade your oscilloscope will be placed in your Tektronix AMS account. An email notification will be sent to your registered mail account. Install the license file to enable the option features.
Includes shipment of calibration data and new front panel bandwidth label.

The option license can only be installed on the oscilloscope for which it was purchased.</td></tr>
<tr><td>SUP5-BW3T56</td><td>Bandwidth upgrade from 350 MHz to 500 MHz on 6-channel models</td></tr>
<tr><td>SUP5-BW3T58</td><td>Bandwidth upgrade from 350 MHz to 500 MHz on 8-channel models</td></tr>
<tr><td>SUP5-BW3T104</td><td>Bandwidth upgrade from 350 MHz to 1 GHz on 4-channel models, plus 4 TPP1000 probes.</td><td rowspan="3">A license file to upgrade your oscilloscope will be placed in your Tektronix AMS account. An email notification will be sent to your registered mail account. Install the license file to enable the option features.
Includes shipment of calibration data and new front panel bandwidth label.

The option license can only be installed on the oscilloscope for which it was purchased.</td></tr>
<tr><td>SUP5-BW3T106</td><td>Bandwidth upgrade from 350 MHz to 1 GHz on 6-channel models, plus 6 TPP1000 probes.</td></tr>
<tr><td>SUP5-BW3T108</td><td>Bandwidth upgrade from 350 MHz to 1 GHz on 8-channel models, plus 8 TPP1000 probes.</td></tr>
</table>

Option name	Description	Notes
SUP5-BW3T204	Bandwidth upgrade from 350 MHz to 2 GHz on 4-channel models, plus 4 TPP1000 probes.	Hardware upgrade; send instrument to Tektronix Service Center. Includes shipment of calibration data and new front panel bandwidth label.
SUP5-BW3T206	Bandwidth upgrade from 350 MHz to 2 GHz on 6-channel models, plus 6 TPP1000 probes.	
SUP5-BW3T208	Bandwidth upgrade from 350 MHz to 2 GHz on 8-channel models, plus 8 TPP1000 probes.	
SUP5-BW5T104	Bandwidth upgrade from 500 MHz to 1 GHz on 4-channel models, plus 4 TPP1000 probes.	Hardware upgrade; send instrument to Tektronix Service Center. Includes shipment of calibration data and new front panel bandwidth label.
SUP5-BW5T106	Bandwidth upgrade from 500 MHz to 1 GHz on 6-channel models, plus 6 TPP1000 probes.	
SUP5-BW5T108	Bandwidth upgrade from 500 MHz to 1 GHz on 8-channel models, plus 8 TPP1000 probes.	
SUP5-BW5T204	Bandwidth upgrade from 500 MHz to 2 GHz on 4-channel models, plus 4 TPP1000 probes.	Hardware upgrade; send instrument to Tektronix Service Center. Includes shipment of calibration data and new front panel bandwidth label.
SUP5-BW5T206	Bandwidth upgrade from 500 MHz to 2 GHz on 6-channel models, plus 6 TPP1000 probes.	
SUP5-BW5T208	Bandwidth upgrade from 500 MHz to 2 GHz on 8-channel models, plus 8 TPP1000 probes.	
SUP5-BW10T204	Bandwidth upgrade from 1 GHz to 2 GHz on 4-channel models, plus 4 TPP1000 probes.	Hardware upgrade; send instrument to Tektronix Service Center. Includes shipment of calibration data and new front panel bandwidth label.
SUP5-BW10T206	Bandwidth upgrade from 1 GHz to 2 GHz on 6-channel models, plus 6 TPP1000 probes.	
SUP5-BW10T208	Bandwidth upgrade from 1 GHz to 2 GHz on 8-channel models, plus 8 TPP1000 probes.	

Record length (options 5-RL-125M, SUP5-RL-125M)

These options let you order the oscilloscope with a larger record length memory, or upgrade a purchased oscilloscope to increase the record length memory. Increased record length memory lets you capture and troubleshoot more waveform data points.

NOTE. *Record length options are not available for the 5 Series MSO Low Profile (MSO58LP).*

Option name	Description
5-RL-125M	Enable 125 million record points per channel maximum. Order when ordering the oscilloscope.
SUP5-RL-125M	Upgrade from 62.5 million record points per channel to 125 million record points per channel maximum. A license file to upgrade your oscilloscope will be placed in your Tektronix AMS account. An email notification will be sent to your registered mail account. Install the license file to enable the option features. The option license can only be installed on the oscilloscope for which it was purchased.

Microsoft Windows 10 operating system (Option 5-WIN or SUP5-WIN)

These options add the Microsoft Windows 10 operating system to your oscilloscope. You can order the option preinstalled at the factory, or purchase and install the solid state drive at a later date. This option is not available for the 5 Series MSO Low Profile (MSO58LP).

Microsoft Windows 10 operating system features

- Provides a Windows environment for loading and running other applications along with running the 5 Series MSO oscilloscope application.
- The 5 Series MSO user interface looks and operates exactly the same as the Linux-based user interface, so you don't have to learn two different versions of the same instrument when you switch from Linux to Windows.

Windows 10 operating system preinstalled option

This option preinstalls this feature when purchasing the oscilloscope.

Install option name	Description
5-WIN	Solid state drive (SSD) with Windows 10 operating system preinstalled on your oscilloscope at the factory.

Windows 10 operating system upgrade option

This option is ordered after purchasing the oscilloscope to install the Windows 10 operating system.

Upgrade option name	Description
SUP5-WIN	A solid state drive with Windows 10 operating system. Simply install the SSD in the drive port located on the bottom of the oscilloscope, and the instrument powers up to Windows 10 operating system and runs the 5 Series MSO as an application. Option SUP5-WIN can be installed on any instrument; it is not tied to a specific instrument.

Arbitrary Function Generator (AFG) (option 5-AFG or upgrade SUP5-AFG)

These options add a 50 MHz AFG function to your oscilloscope. You can order the option preinstalled at the factory, or purchase and install an option license at a later date.

AFG features

- Function types: Arbitrary, Sine, Square, Pulse, Ramp, Triangle, DC Level, Gaussian, Lorentz, Exponential Rise/Fall, Sin(x)/x, Random Noise, Haversine, Cardiac
- Maximum frequency: 50 MHz (Sine)
- Maximum output amplitude: 5 Vp-p
- Maximum sample rate: 250 MS/s
- Arbitrary function record length: 128K samples

AFG preinstalled option

This option preinstalls this feature when purchasing the oscilloscope.

Install option name	Description
5-AFG	AFG feature preinstalled on your oscilloscope at the factory. Adds arbitrary and function generation capability to your instrument.

AFG upgrade option

This option is ordered after purchasing the oscilloscope to enable the AFG feature.

Upgrade option name	Description
SUP5-AFG	Adds arbitrary and function generation capability to your instrument. A license file to upgrade your oscilloscope will be placed in your Tektronix AMS account. An email notification will be sent to your registered mail account. Install the license file to enable the option features. The option license can only be installed on the oscilloscope for which it was purchased.

Enhanced instrument security (Option 5-SEC)

Advanced instrument security option 5-SEC provides the highest level of instrument security for 5 Series MSO products. The oscilloscope hardware is configured to easily declassify the oscilloscope. Option 5-SEC must be ordered at the same time you order an instrument.

Enhanced instrument security preinstalled option

This option preinstalls this feature when ordering the oscilloscope. Option 5-SEC must be ordered at the same time you order an instrument.

Install option name	Description
5-SEC	No user access to the internal m.2 storage to store or save any user data to the oscilloscope memory. Data can only be saved to or read from a USB storage device connected to the instrument, or through the programmable interface. No hard drive connection bracket on the bottom of the instrument prevents installing the optional 5-WIN Windows OS solid state drive. Password protection to enable/disable external USB Host, USB Device and Ethernet communication ports. Password protection to enable/disable firmware upgrades or downgrades

Advanced instrument security upgrade option

You cannot order this option as a field-installable upgrade, as the option requires hardware reconfiguration.

Serial bus and trigger options

Serial bus and trigger options provide bus display and triggering for testing and analysis of industry standard serial buses. You can order the oscilloscope with options preinstalled, or upgrade a purchased oscilloscope to add these options.

Factory-installed serial trigger options

These options preinstall the feature when purchasing the oscilloscope.

Option	Description
5-SRAERO	Aerospace serial triggering and analysis (ARINC 429, MIL-STD-1553)
5-SRAUDIO	Audio (I^2S, LJ, RJ, TDM)
5-SRAUTO	Automotive serial triggering and analysis (CAN, CAN FD, LIN, FlexRay)
5-SRCOMP	Computer serial triggering and analysis (RS-232/422/485/UART)
5-SRENET	Ethernet serial triggering and analysis (10BASE-T, 100BASE-T)
5-SREMBD	Embedded serial triggering and analysis (I^2C, SPI)
5-SRUSB2	USB serial triggering and analysis (USB 2.0 LS, FS, HS)

Serial trigger upgrade options

These options can be ordered and installed on already purchased oscilloscopes.

<table>
<tr><th>Option name</th><th>Description</th><th>Notes</th></tr>
<tr><td>SUP5-SRAERO</td><td>Aerospace serial triggering and analysis (ARINC 429, MIL-STD-1553)</td><td rowspan="7">A license file to upgrade your oscilloscope will be placed in your Tektronix AMS account. An email notification will be sent to your registered mail account. Install the license file to enable the option features.
The option license can only be installed on the oscilloscope for which it was purchased.</td></tr>
<tr><td>SUP5-SRAUDIO</td><td>Audio serial triggering and analysis (I^2S, LJ, RJ, TDM)</td></tr>
<tr><td>SUP5-SRAUTO</td><td>Automotive serial triggering and analysis (CAN, CAN FD, LIN, FlexRay)</td></tr>
<tr><td>SUP5-SRCOMP</td><td>Computer serial triggering and analysis (RS-232/422/485/UART)</td></tr>
<tr><td>SUP5-SRENET</td><td>Ethernet serial triggering and analysis (10BASE-T, 100BASE-T)</td></tr>
<tr><td>SUP5-SREMBD</td><td>Embedded serial triggering and analysis (I^2C, SPI)</td></tr>
<tr><td>SUP5-SRUSB2</td><td>USB serial triggering and analysis (USB 2.0 LS, FS, HS)</td></tr>
</table>

Compliance testing options

Compliance testing options provide compliance testing and analysis of industry standard buses. You can order the oscilloscope with options preinstalled, or upgrade a purchased oscilloscope to add these options.

NOTE. *Compliance testing options are not available for the 5 Series MSO Low Profile (MSO58LP).*

Factory-installed compliance testing options

This option preinstalls the feature when purchasing the oscilloscope.

Option	Description
5-CMAUTOEN	Automotive Ethernet (100BASE-T1) compliance testing. Requires option 5-WIN or SUP5-WIN (SSD with Microsoft Windows 10 operating system).

Compliance testing upgrade options

This option can be ordered and installed on already purchased oscilloscopes.

Option name	Description	Notes
SUP5-CMAUTOEN	Automotive Ethernet (100BASE-T1) compliance testing. Requires option 5-WIN or SUP5-WIN (SSD with Microsoft Windows 10 operating system)	A license file to upgrade your oscilloscope will be placed in your Tektronix AMS account. An email notification will be sent to your registered mail account. Install the license file to enable the option features. The option license can only be installed on the oscilloscope for which it was purchased.

Advanced Jitter Analysis (options 5-DJA, SUP5-DJA)

These options add over 30 industry standard automatic jitter and eye diagram measurements, as well as a jitter summary table with the most common measurements. You can order the option preinstalled at the factory, or purchase and install an option license at a later date.

Advanced Jitter Analysis preinstalled option

This option preinstalls this feature when ordering the oscilloscope.

Install option name	Description
5-DJA	Advanced jitter and eye analysis preinstalled on your oscilloscope at the factory. Adds over 30 Jitter and Eye measurements, including TIE, TJ@BER, DDJ, DCD, SRJ, Eye Height, Eye Width, Eye High and Low, Q-Factor, Differential Crossover, T/nT Ratio, SSC Frequency deviation, SSC Modulation Rate, and more.

Advanced Jitter Analysis upgrade option

This option can be ordered after purchasing the oscilloscope.

Upgrade option name	Description
SUP5-DJA	Adds over 30 Jitter and Eye measurements, including TIE, TJ@BER, DDJ, DCD, SRJ, Eye Height, Eye Width, Eye High and Low, Q-Factor, Differential Crossover, T/nT Ratio, SSC Frequency deviation, SSC Modulation Rate, and more. A license file to upgrade your oscilloscope will be placed in your Tektronix AMS account. An email notification will be sent to your registered mail account. Install the license file to enable the option features. The option license can only be installed on the oscilloscope for which it was purchased.

Advanced Power Analysis (Option 5-PWR, SUP5-PWR, or 5-PS2)

This option adds industry standard power measurements, as well as a summary table that displays the most common measurements. You can order the option pre-installed at the factory, or purchase and install an option license at a later date.

Power pre-installed options

Order these options to pre-install power measurement options on your oscilloscope at the factory.

Install option name	Description
5-PWR	Power measurements are pre-installed on your oscilloscope at the factory. Power measurements include Harmonics, Power Quality, Line Ripple, Switching Ripple, SOA, di/dt, dv/dt, Switching Loss, Amplitude, Cycle Maximum, Cycle Minimum, Frequency and more.
5-PS2	Power measurement bundle. Adds the power measurements, including Harmonics, Power Quality, Line Ripple, Switching Ripple, SOA, di/dt, dv/dt, Switching Loss, Amplitude, Cycle Maximum, Cycle Minimum, Frequency and more. Also included probes THDP0200, TCP0030A, and the Power Deskew and Calibration fixture (067-1686-XX). Can only be ordered with factory purchase of instrument.

Power upgrade option

Order this option to upgrade your oscilloscope with power measurements.

Upgrade option name	Description
SUP5-PWR	Adds the advanced power measurements, including Harmonics, Power Quality, Line Ripple, Switching Ripple, SOA, di/dt, dv/dt, Switching Loss, Cycle Amplitude, Cycle Maximum, Cycle Minimum, Frequency and more. A license file to upgrade your oscilloscope will be placed in your Tektronix AMS account. An email notification will be sent to your registered mail account. Install the license file to enable the option features. The option license can only be installed on the oscilloscope for which it was purchased.

Power cord options

These options let you order the oscilloscope with a country- or region-specific power cord.

Power cord options

These options are ordered when ordering the oscilloscope.

Option name	Description
A0	North America Power Cord
A1	Universal EURO Power Cord
A2	United Kingdom Power Cord
A3	Australia Power Cord
A5	Switzerland Power Cord
A6	Japan Power Cord
A10	China Power Cord
A11	India Power Cord
A12	Brazil Power Cord
A99	No Power Cord or AC Adapter
E1	Standard user manual with UK, Universal EURO, and Swiss power cords

Service options

Service options improve the level of service response, extend the warranty period, or provide calibration reports. You can order service options when you purchase an oscilloscope, or purchase a service option at a later date.

Service options

Option name	Description
T3	Three year Total Protection Plan. Includes preventative maintenance, and repair or replacement coverage from wear and tear, accidental damage, and ESD or EOS damage. Includes a five-day turnaround time and priority access to customer support.
T5	Five year Total Protection Plan. Includes preventative maintenance, and repair or replacement coverage from wear and tear, accidental damage, and ESD or EOS damage. Includes a five-day turnaround time and priority access to customer support.
R5	Standard warranty is extended to five years. Covers parts, labor, and 2-day shipping within country. Guarantees faster repair time than without coverage. All repairs include calibration and updates. Hassle free - a single call starts the process.
C3	Three-year calibration service. Includes traceable or functional verification where applicable, for recommended calibrations. Coverage includes the initial calibration plus two years calibration coverage.
C5	Five-year calibration service. Includes traceable or functional verification where applicable, for recommended calibrations. Coverage includes the initial calibration plus four years calibration coverage.
D1	Factory calibration data report for the instrument.
D3	Three years of calibration data reports (with Option C3)

Option name	Description
D5	Five years of calibration data reports (with Option C5)
R5DW	Repair service coverage for five years (includes the standard product warranty period). The five year period starts at time of instrument purchase.

How to install an option license

Use this process install an option license to enable specific instrument features. Options provide advanced functions for specific standards or testing requirements.

Prerequisites:

- A license file for each option. Contact Tektronix Customer Service to purchase and obtain option license file(s). Option licenses are specific to the oscilloscope serial number, and cannot be used on other oscilloscopes.

  License (.lic) files are downloaded from the Tektronix AMS tool at www.tek.com/products/product-license. There is a how-to video on this site to help you with your license file install.
- To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

NOTE. *You can only install an option license one time. If you need to reinstall an uninstalled license, contact Tektronix Customer Support.*

---

1. Copy the option license file (<filename>.lic) onto a USB memory device.
2. Insert the USB memory device into the oscilloscope.
3. Select **Help > About**.
4. Tap **Install License**.
5. Navigate to and select the license file (<filename>.lic) on the USB memory device. See the table at the end of this procedure.
6. Tap **Open**. The oscilloscope enables the option license and returns to the About screen. Verify that the installed option license is in the list.
7. Power cycle the oscilloscope before taking any measurements.

   To uninstall (return) a license, see *How to uninstall (return) an option license* on page 16.

Drive names for USB memory devices.

How to uninstall (return) an option license

Use this process to uninstall an option license.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

NOTE. *You cannot reinstall an option license once you have uninstalled it. If you need to reinstall an uninstalled license, contact Tektronix Customer Support to obtain a new license file.*

---

1. Select **Help > About**.
2. Tap the option license in the list that you want to uninstall.
3. Tap the **Return License** button. The oscilloscope opens the **Location to Save the Exit Key** menu.
4. Navigate to and select the location to save the option license key uninstall file. See the table at the end of this procedure.

5. Tap **Create.** The oscilloscope saves the license key to the specified location and removes the license from the Installed Options list.
6. Power cycle the oscilloscope before taking any measurements.

   To install an option license, see *How to install an option license* on page 15.

Drive names for system and USB memory devices.

Use the table to determine which drive to select when navigating to and/or selecting a file on system memory or a connected USB memory device.

Drive name	Drive letter	Drive or physical USB port location
MSO54, MSO56, MSO58		
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**E**	USB 3.0 (left)
	F	USB 2.0 (center)
	G	USB 2.0 (right)
Rear panel	**H**	USB 2.0 (top)
	I	USB 2.0 (bottom)
	J	USB 3.0 (top)
	K	USB 3.0 (bottom)
MSO58LP		
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**E**	USB 3.0 (left)
	F	USB 2.0 (right)
Rear panel	**G**	USB 2.0 (top)
	H	USB 2.0 (bottom)
	I	USB 3.0 (top)
	J	USB 3.0 (bottom)
Option 5-WIN, SUP5-WIN instruments and USB port labels		
Root drive	**C**	User-accessible memory on the oscilloscope.
USB ports	**Dynamic port letter assignment**	If option 5-WIN or SUP5-WIN is installed (Windows operating system), the Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. Use standard Windows procedures to mount and access network drives.

Install your instrument

Check shipped accessories

Make sure that you received everything you ordered. If anything is missing, contact Tektronix Customer Support. In North America, call 1-800-833-9200. Worldwide, visit www.tek.com to find contacts in your area.

Check the packing list that came with your instrument to verify that you have received all ordered items. Check that you received all standard accessories.

MSO54, MSO56, MSO58

Item	Quantity	Tektronix part number
5 Series MSO (MSO54, MSO56, MSO58) Installation and Safety Manual	1	071-3514-xx
TPP0500B Passive Voltage Probe (500 MHz bandwidth). Shipped with 350 MHz and 500 MHz models.	One per channel	TPP0500B
TPP1000 Passive Voltage Probe (1 GHz bandwidth). Shipped with 1 GHz and 2 GHz models.	One per channel	TPP1000
Front cover	1	200-5406-xx
Accessory pouch (attached to front cover)	1	016-2106-xx
Mouse (wired with USB connector)	1	119-7054-xx
Power cord	1	Depends on region
Calibration certificate	1	N/A
Report of factory installed licenses	1	N/A

MSO58LP

Item	Quantity	Tektronix part number
5 Series MSO MSO58LP Installation and Safety Manual	1	071-3568-xx
Power cord	1	Depends on region
Calibration certificate	1	N/A
Report of factory installed licenses	1	N/A

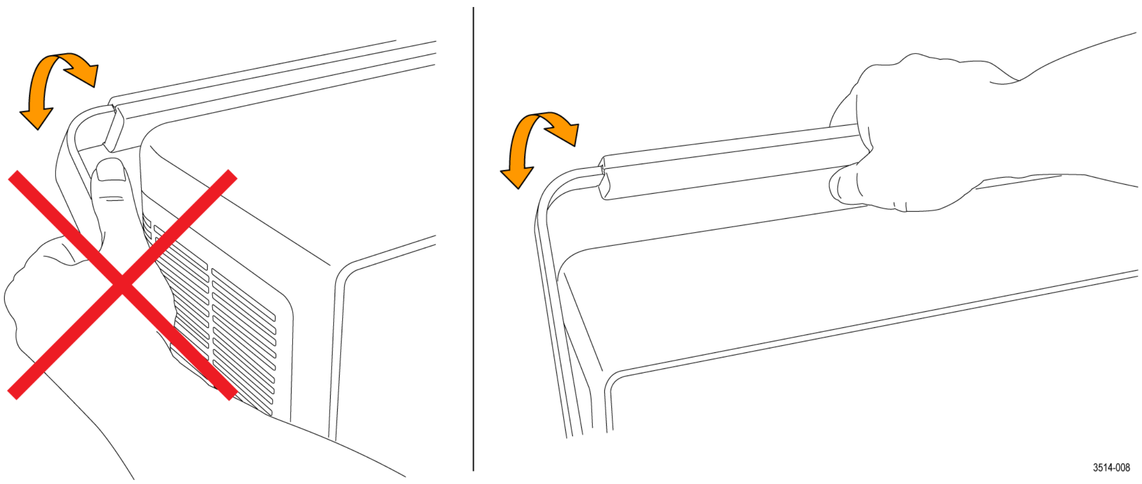
If you purchased factory installed options such as Advanced Serial Triggering, tap **Help > About** to confirm that the option(s) are listed in the **Installed Options** table.

Safely rotate the handle

Use the correct process to eliminate the chance of pinching your thumb or rear-panel-connected cables while rotating the handle. This does not apply to MSO58LP instruments.

CAUTION. *Hold the top of the handle to rotate the handle on the instrument. Do not hold the handle from the sides and rotate, as this can pinch the base of your thumb between the handle and the case.*

If you have routed any cables between the handle and the case, be careful when rotating the handle so that you do not pinch the cables.



Operating requirements

Use the oscilloscope 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 +122 °F) For proper cooling, keep the sides and rear of the instrument clear of obstructions for 2 inches (51 mm).
Operating humidity	5% to 90% relative humidity (% RH) up to +40 °C (+104 °F) 5% to 55% RH above +40 °C up to +50 °C (+104 °F to +122 °F), Noncondensing.
Operating altitude	Up to 3000 meters (9842 feet)

Power requirements

Characteristic	Description
Power source voltage	100 V - 240 $V_{AC\ RMS}$, ±10%, single phase
Power source frequency	50/60 Hz, 100-240 V (90-264 V) 400 Hz, 115 V (103-127 V)
Power consumption	All models: 400 W maximum

Input signal requirements

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

Make sure that input signals are within the following requirements.

Input	Description
Analog input channels, 1 M Ω setting, maximum input voltage at BNC	300 V_{RMS} Measurement Category II
Analog input channels, 50 Ω setting, maximum input voltage at BNC	5 V_{RMS}
Digital input channels, maximum input voltage range at digital inputs	Observe probe ratings TLP058; ±42 V_P
Ref In maximum input voltage at BNC (rear panel)	7 V_{PP}

Powering the oscilloscope

Use this procedure to connect the oscilloscope to line power and power on and off the oscilloscope. Always connect the oscilloscope to AC power using the power cord that shipped with the instrument.

Prerequisite: Use the AC power cord that shipped with your oscilloscope.

1. Connect the supplied power cord to the oscilloscope power connector.

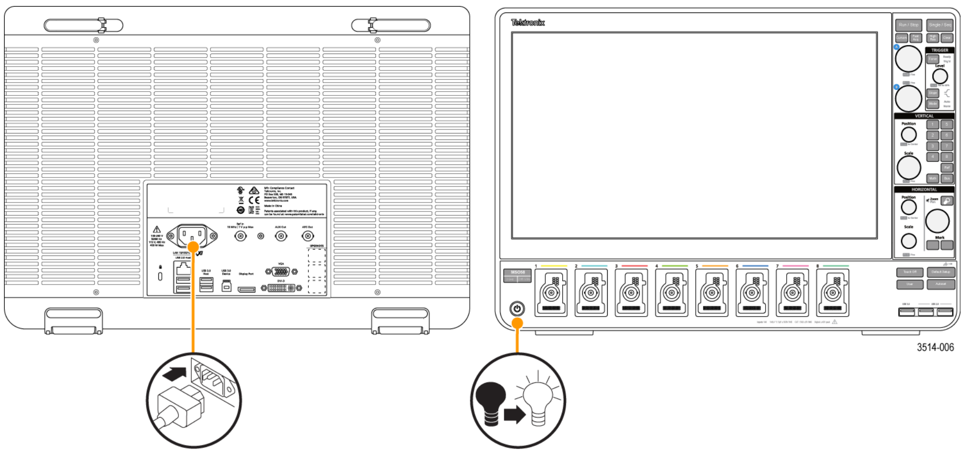


Figure 1: MSO54, MSO56, MSO58

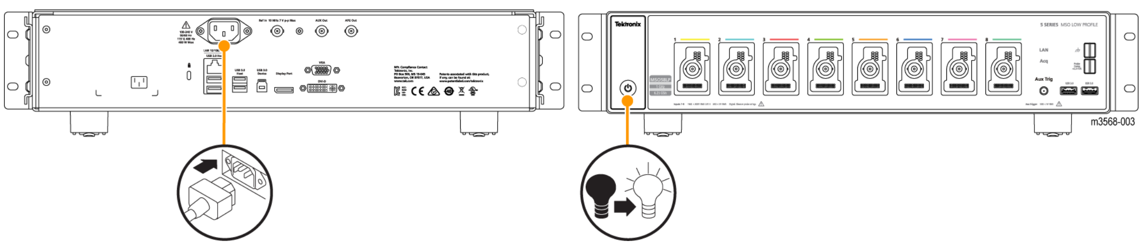


Figure 2: MSO58LP

2. Connect the power cord to an appropriate AC mains source.

   Power is supplied to the power supply and some other boards whenever the AC power cord is connected to a live mains circuit, putting the instrument in standby mode.

3. Push the front panel power button to power the instrument on and off.

   The power button color indicates instrument power states:

   No light – no AC power applied

   Yellow – standby mode

   Blue – powered on

4. To completely remove power from the instrument, disconnect the power cord.
5. To transport the instrument with its power cord, flip out the power cord supports on the upper edge of the rear panel and wrap the power cord around the supports.

Check that oscilloscope passes power-on self tests

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

Prerequisite:

For MSO58LP, connect a monitor to a video output on the rear panel, and connect a mouse to a USB port.

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

   If the power-on self test shows **Failed**:

   a. Power cycle the oscilloscope.

   b. Tap **Utility > Self Test**. If the power-on self test still shows **Failed**, contact Tektronix Customer Support.

Installing and activating Windows 10 (option 5-WIN, SUP5-WIN)

Use the following instructions to install the option SUP5-WIN Microsoft Windows 10 solid state drive (upgrade installed after purchasing the instrument) and activate Windows 10 (5-WIN, SUP5-WIN) on MSO54, MSO56, and MSO58 instruments.

NOTE. *Option 5-WIN and SUP5-WIN cannot be installed in the 5 Series MSO Low Profile (MSO58LP) instrument.*

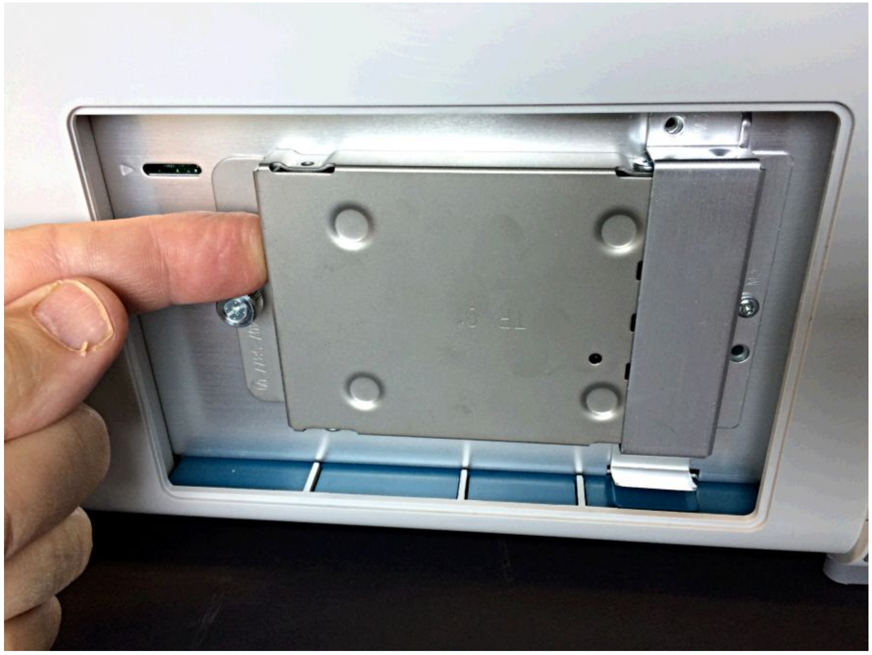
Install the SUP5-WIN SSD drive

The option SUP5-WIN SSD assembly installs in the bottom of the 5 Series MSO instruments.

Prerequisite: Wear an anti-static wrist strap connected to the instrument chassis while installing this drive.

1. Remove all cables from the front and rear of the instrument, including the power cable.
2. Position the instrument on its back, with the bottom facing you.
3. Remove the SSD drive cover from the bottom of the oscilloscope as shown.

4. Slide the connector end of the SSD into the drive connector bracket. Push firmly to seat the drive in the connector.



5. Push down on and tighten the thumb screw to attach the SSD drive to the chassis.
6. Reinstall the SSD drive cover on the bottom of the instrument.
7. Restore the instrument to its normal operating position.
8. Go to the *Windows OS power on* procedure.

Powering on Windows for the first time

The instrument goes through a series of configuration bootups that require manual intervention when first powering up a new (never-installed) SUP5-WIN drive in an instrument. Once the SUP5-WIN drive is configured, the instrument boots into Windows and starts the oscilloscope application from then on.

1. Power on the instrument. The oscilloscope powers up to initialize some settings, then shuts down.
2. Power on the instrument again. The instrument powers up into the application startup screen, then displays a message at the bottom of the screen:

   Updating System Files... This will take a few minutes.

   The oscilloscope will shut down when the update is complete.

   After a few minutes, the instrument powers down.
3. Power on the instrument again. The instrument boots up into Windows and starts the oscilloscope application.

The instrument attempts to activate the Windows license as part of the initial power-on process (See *Activating Windows* on page 25).

Activating Windows

Activating Windows. The Windows operating system shipped from Tektronix is in a "deferred activation" state. The first time you power on an instrument with a newly installed SUP5-WIN drive, the operating system may attempt to activate itself, depending on whether the instrument is connected to a network.

- The instrument is connected to a network with access to the Microsoft web site:
  - The Windows activation occurs silently in the background, and does not display any messages. No other action is required.
  - See *Verifying Windows activation* on page 25 to check the activation status of the instrument.
- The instrument is connected to a network without access to the Microsoft web site:
  - The instrument may attempt to activate and fail. You may see a screen message saying that Windows is not activated and the Windows user settings may be disabled.
  - See *Verifying Windows activation* on page 25 to check the activation status of the instrument.
  - You can activate Windows by either connecting the instrument to a network with access to Microsoft or by contacting Microsoft to obtain activation instructions:
    - Go to **https://support.microsoft.com/en-us** and select **Windows.**
    - Go to **https://support.microsoft.com/en-us/contactus/** and follow the instructions to select your preferred contact method.

  ***NOTE.** Please contact Microsoft to resolve Windows activation issues.*

- The instrument is not connected to any network:
  - The instrument stays in the "deferred activation" state. No error message appears. Windows, and the oscilloscope application, can operate indefinitely in the deferred activation state without error messages.
  - See *Verifying Windows activation* on page 25 to check the activation status of the instrument.
  - Once you connect the instrument to a network with access to Microsoft, Windows will activate automatically (no interaction required).
- Moving an activated SUP5-WIN SSD from one instrument to another:
  - Windows should remain activated. To verify activation, see *Verifying Windows activation* on page 25. If Windows lost activation when installed on the current instrument, follow the above instructions to re-activate.

Verifying Windows activation. To check that Windows is activated:

1. Tap **Start** on the Windows taskbar.
2. Scroll down and tap **Settings**.
3. Tap **Update & Security**.
4. Tap **Activation** (left side list) to display the activation status.

Windows update. Automatic Windows update is disabled by default.

Differences between Windows and base instrument user interfaces

The Windows-based oscilloscope application interface appearance and behavior is exactly the same as the base 5 Series MSO instruments, with some exceptions:

- You can change the size of the Windows-based oscilloscope application, or minimize it, just like any other Windows application.
- The File Utilities choice in the File menu opens a standard Windows Explorer instance instead of the custom dialog found in the base instrument.
- You can use the standard Windows network tools to mount and access network drives.
- You can use the standard Windows tools to create a login password to access the instrument, if required by your organization.

Updating the Windows TekScope application software

The 5 Series MSO Windows TekScope application does not automatically update when newer versions are released. You will need to manually download and install the newer software.

To update the Windows version of the 5 series MSO oscilloscope application software:

Instrument is connected to a network:

1. Display the Windows desktop and open a browser.
2. Go to **www.tek.com/product-support**.
3. Enter your model number in the **Enter Product or Product Series Name:** field and click **Go**.
4. Click the **Software** tab (left side of table).
5. Click the link in the description that pertains to your instrument model or series, and select the software that applies for Windows instruments.
6. Check that the listed software is newer than your current installed software before downloading and installing.
7. Follow instructions to download the application file.
8. Click on the install instructions link to open the installation instructions.

Instrument is not connected to a network:

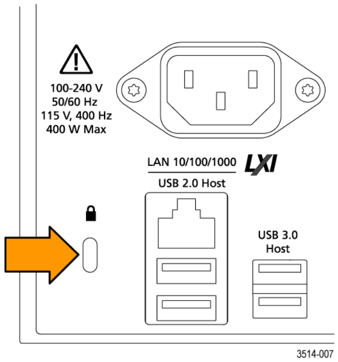
Follow the above instructions on a network-connected PC,

1. Open a browser on a network-connected PC or laptop.
2. Do steps 2 through 7 above.
3. Download the installation file to a USB memory device and insert the USB memory device in any USB Host port on the instrument.
4. Open the USB drive location and copy the executable file to the desktop.
5. Double-tap on the installation file to update the application software; follow any on-screen instructions.

Secure (lock) the oscilloscope

Lock an oscilloscope to a test bench or equipment rack to prevent property loss.

Attach a standard laptop security lock to the rear panel of the oscilloscope, to secure the oscilloscope to a workbench, rack, or other location.



Connecting Probes

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

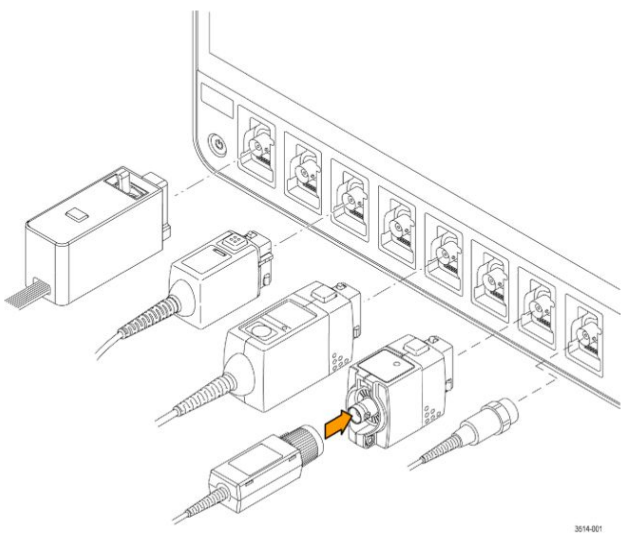


Figure 3: MSO54, MSO56, MSO58

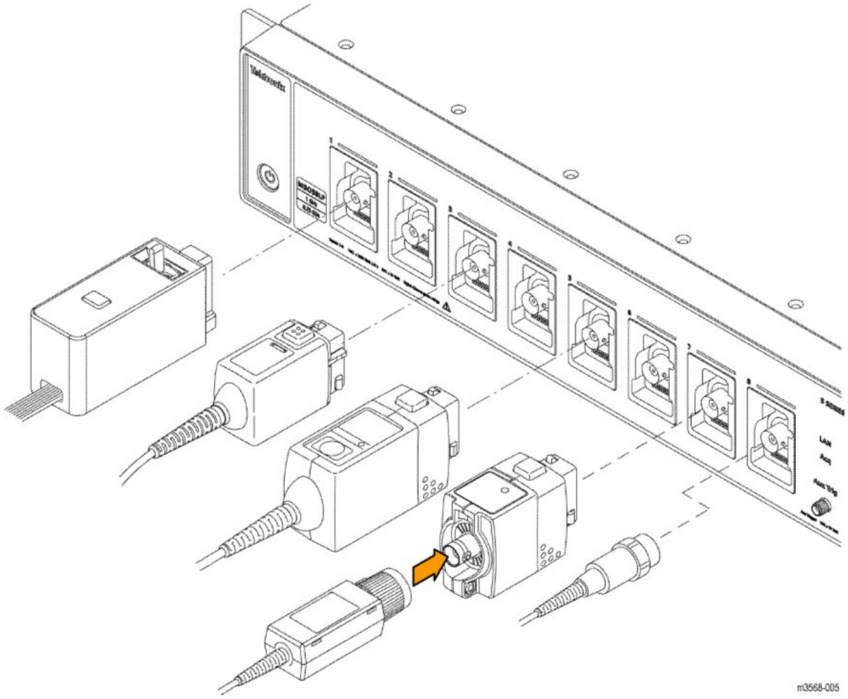


Figure 4: MSO58LP

Connect TPP0500, TPP1000, TekVPI+®, TekVPI®, or other supported Tektronix analog probes by pushing them into a FlexChannel connector. The probe base latch locks with a 'click' when the probe is fully seated.

TekVPI probes automatically set the channel input parameters for that probe (bandwidth, attenuation, termination, and so on). If a probe has a **Menu** button, push that button to open an on-screen configuration menu. Follow instructions provided with active probes to set their parameters (auto zero, degauss, and so on).

To connect a TLP058 FlexChannel Logic Probe:

1. Move the locking lever to the unlocked position, then let go to reset locking lever to the center position.
2. Insert the probe into a FlexChannel channel until fully seated and the lock mechanism clicks.
3. Move the locking lever to the locked position. The status light should be a solid green.
4. To disconnect the TLP058 probe, move and hold the locking lever at the unlocked position and pull out the probe. Do not pull on the ribbon cable while removing the probe.

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

Connecting a probe does not automatically display or enable that channel. Tap an Inactive Channel button to turn on that channel. Double-tap the Channel badge to open its configuration menu to verify or change probe or cable settings (bandwidth, attenuation, termination and so on).

Rackmount information

The optional RM5 Rackmount Kit for the MSO54, MSO56, and MSO58 lets you install the oscilloscope in standard equipment racks. The rack mount requires seven rack units (7U) of space to install. The MSO58LP ships with attached rack mount hardware to install into a 2U rack space.

MSO54, MSO56, and MSO58

Contact Tektronix Customer Support to purchase the rackmount kit option RM5. Follow the instructions that came with the rackmount kit (*RM5 Rackmount Kit Instructions*, Tektronix part number 071-3523-xx).

Make sure to allow adequate clearance on the sides and rear for air ventilation, and on the back for any cables you attach to the rear panel.

MSO58LP

The MSO58LP comes equipped with rack mount hardware installed so that you can add it to a rack. Follow the instructions in the 5 Series MSO MSO58LP Installation and Safety Manual to install the instrument in a rack.

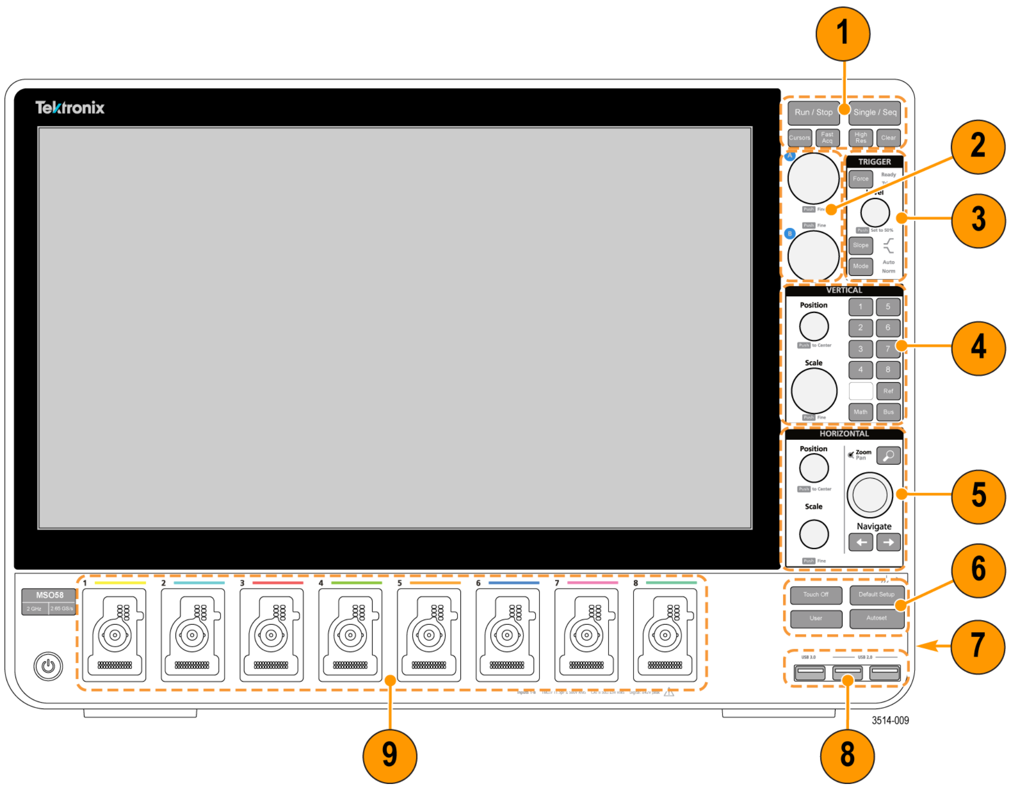
To use an MSO58LP on a bench, purchase and install the 5 Series MSO MSO58LP Benchtop Conversion kit (Tektronix part number 020-3180-xx). The kit includes chassis feet and a handle, and lets you stack instruments on a bench.

Getting acquainted with your instrument

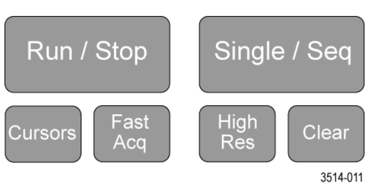
The following content provides a high-level description of the instrument controls and user interface.

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, or insert USB devices.



1. **Acquisition** and **Cursors** controls:

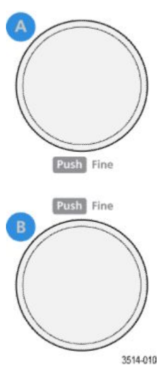


- **Run/Stop** starts and stops waveform acquisition. The button color indicates the acquisition status (green = running and acquiring; red = stopped). When stopped, the oscilloscope shows waveforms from the last completed acquisition. The Run/Stop button on the screen also shows the acquisition status.
- **Cursors** button turns screen cursors on or off. Use the Multipurpose knobs to move the cursors. Double-tap the cursor readouts, or on a cursor bar (line), to open the configuration menu to set cursor types and functionality. See *Display and configure cursors* on page 89.

- **Fast Acq™** enables or disables the fast acquisition mode. FastAcq provides high-speed waveform capture that reduces the dead time between waveform acquisitions, enabling the capture and display of transient events such as glitches and runt pulses. It is helpful in finding elusive signal anomalies. Fast acquisition mode can also display waveform phenomena at an intensity that reflects their rate of occurrence.
- **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 = single acquisition acquired; solid green = waiting for trigger event). Pushing **Single/Seq** again takes another single acquisition.
- **High Res** applies unique finite impulse response (FIR) filters based on the current sample rate. This FIR filter maintains the maximum bandwidth possible for that sample rate while rejecting aliasing. The filter removes noise from the oscilloscope amplifiers and ADC above the usable bandwidth for the selected sample rate. Implementation of the filter in hardware, ahead of the trigger and storage, reduces trigger jitter and enables using **Fast Acq** mode while in **High Res** mode.

  High Res mode also guarantees at least 12 bits of vertical resolution. The number of bits of resolution is displayed in the **Acquisition** badge at the bottom of the screen. The **Horizontal** badge also updates to show the sample rate and record length settings while in **High Res** mode.
- **Clear** deletes the current acquisitions and measurement values from memory.

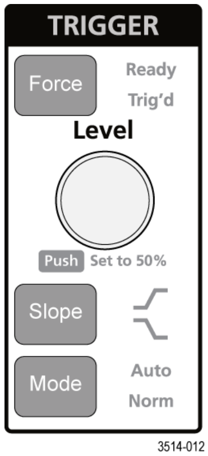
2. **Multipurpose knobs**:



- **Multipurpose knobs (A, B)** The Multipurpose knobs A and B move cursors and set parameter values in configuration menu input boxes. Selecting an input box that can use a Multipurpose knob assigns the indicated knob to change the value in that input box. The ring 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 exit **Fine** mode.

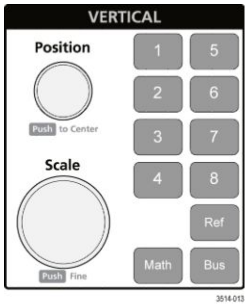
3. Trigger controls:



- **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 indicates the trigger source except for dual-level triggers. The **Level** knob is disabled 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 (low to high, high to low, or either direction). Push the button to cycle through the selections. The **Slope** button is disabled when the trigger type requires other slope qualifiers (set from the **Trigger** configuration menu).
- **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.

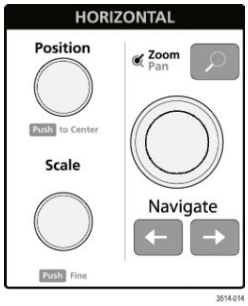
  Normal trigger mode sets the instrument to acquire and display a waveform only when there is a valid trigger event. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.

4. Vertical controls:



- **Position** moves the selected waveform (Channel, Math, Reference, Bus) and its graticule up or down on the screen. The color of the **Position** 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.
- **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, and are specific to the selected waveform in both **Stacked** or **Overlay** modes (in other words, each waveform has its own unique vertical graticule settings regardless of display mode). The color of the **Scale** knob indicates which waveform the knob is controlling.
- **Channel** buttons (1-4 for MSO54, 1-6 for MSO56, 1-8 for MSO58) turns on (displays), selects, or turns off a channel, as follows:
  - 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).
- The **Math** button adds or selects a Math waveform on the Waveform view, as follows:
  - 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.
- The **Ref** button adds or selects a Reference (saved) waveform on the Waveform view, as follows:
  - If no Reference waveform exists, pushing the **Ref** 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.
  - If only one Reference waveform is displayed, pushing the button turns off the Reference waveform (removes it from the Waveform View). Push the button again to display the waveform.
  - If two or more Reference waveforms are displayed, pushing the button cycles through selecting each Reference waveform.
- The **Bus** button adds or selects a bus waveform on the Waveform view, as follows:
  - If no Bus waveform exists, pushing the **Bus** 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.

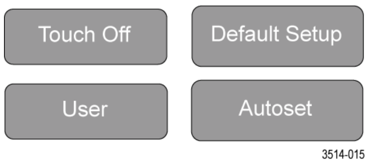
5. **Horizontal** controls:



- **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.
- **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 exit Fine mode.
- **Zoom** opens the Zoom mode. Push **Zoom** again to exit zoom mode. See *The Zoom user interface elements* on page 52.
- **Zoom** knob (ce9nter knob) increases or decreases the area of the zoom box in the Zoom Waveform Overview, which in turn controls the zoom amount of the waveforms shown in the main Zoom view.
- **Pan** knob (outer knob) moves the Zoom box left or right in the **Zoom Waveform Overview**, which in turn controls the part of the waveform shown in the main Zoom view.
- **Navigate** (left and right arrow) buttons puts the oscilloscope in Zoom mode and positions the previous or next search point in the waveform record to the center graticule of the Waveform view. There must be a **Search** badge present in the Results bar before the **Navigate** function will operate. See *Badges* on page 45.

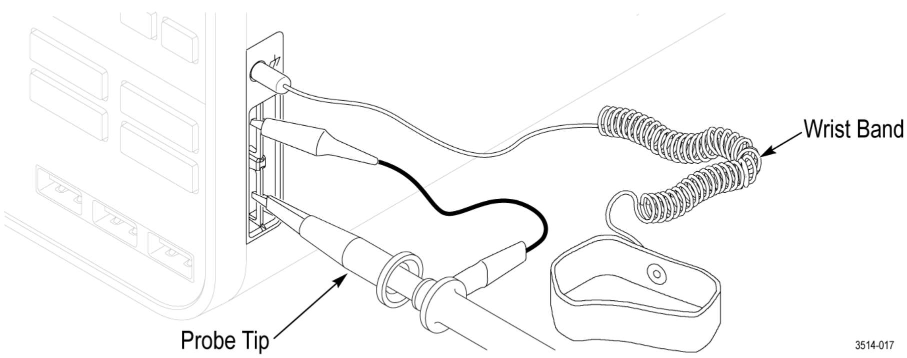
  The front panel **Navigate** buttons can also be used for the **Previous** and **Next** button functions on measurement badges.

6. Miscellaneous controls:



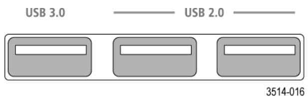
- **Touch Off** turns touch screen capability off. The **Touch Off** button is lighted when the touch screen is turned off.
- **User** is a one-push save operation that uses the current menu bar **File > Save As** settings to save screen shots (including open menus and dialog boxes), waveform files, instrument settings, and so on, as follows:
  - If a **File > Save** or **File > Save As** operation has occurred since the last instrument startup, pushing **User** 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 **User** opens the **Save As** configuration menu. Select a tab to select what type of file to save (Screen Capture, Waveform, and so on), set any associated parameters, and where to save it, and select **OK**. The specified file or files are saved. The next time you push **User**, the specified files are saved.
  - Screen Captures capture the entire screen, including most displayed configuration menus and dialog boxes.
- **Default Setup** restores the oscilloscope settings (horizontal, vertical, scale, position, and so on) to the factory default settings.
- **Autoset** automatically displays a stable waveform. See *Quickly display a waveform (Autoset)* on page 67.

7. Ground and Probe Compensation connectors:



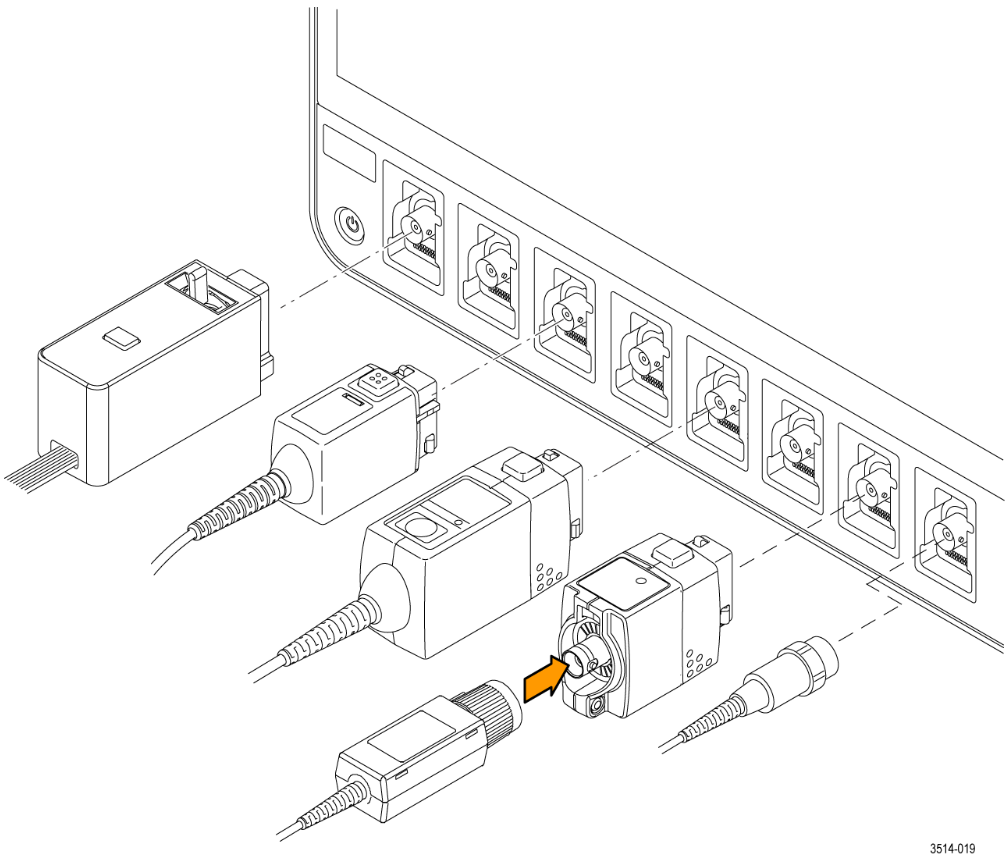
- The Ground and Probe Compensation connectors are located at the lower right side of the instrument, near the front panel. The Ground connector (the small hole in the case) provides an electrically grounded (through a resistor) connection point to attach an anti-static wrist strap, to reduce electrostatic damage (ESD) while you handle or probe the DUT.
- The Probe Compensation connections provide a ground connector (upper tab) and 1 kHz square wave source (lower tab) for adjusting the high-frequency response of a passive probe (probe compensation). The oscilloscope uses this signal to automatically compensate supported probes, including the ones that ship with the product. See *Compensate the TPP0500B or TPP1000 probes* on page 59.

8. USB Host ports (USB 3.0 and 2.0):



- USB ports are located at the lower right corner of the front panel, and on the rear panel. Connect USB flash drives to which you can save or recall data (such as instrument software updates, waveforms, settings, and screen captures), or connect peripheral devices such as a mouse or keyboard.

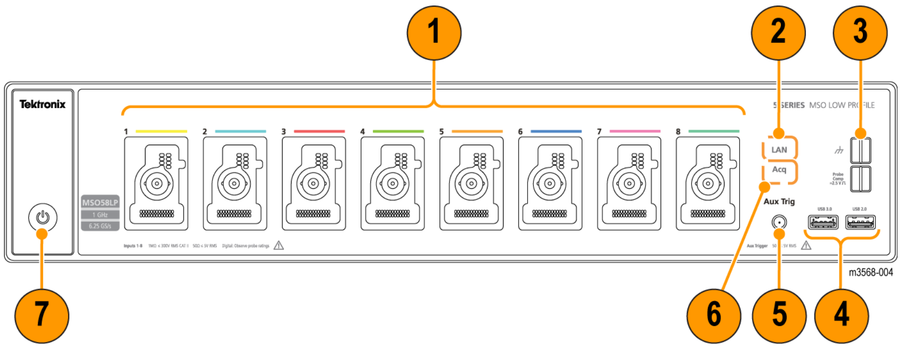
9. FlexChannel probe connectors:



- FlexChannel connectors support all TekVPI+ and TekVPI measurement probes, BNC passive probes, the TPL058 FlexChannel Logic Probe, and BNC cables. You connect most probes simply by pushing them into the connector until the probe seats with a click. See *Connecting Probes* on page 27.

MSO58LP front panel connections

The front panel connections are where you power on or off the instrument, input signals with probes or cables, connect an external trigger input signal, insert USB devices, and access a probe compensation signal.



1. **FlexChannel probe connectors**:

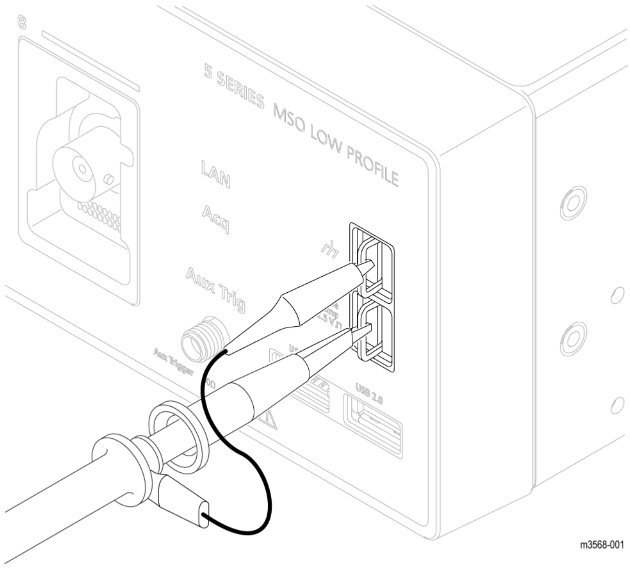
   FlexChannel connectors support all TekVPI+ and TekVPI measurement probes, BNC passive probes, the TPL058 FlexChannel Logic Probe, and BNC cables. You connect most probes simply by pushing them into the connector until the probe seats with a click. See *Connecting Probes* on page 27

2. **LAN** Status LED:

   Shows the network connection and activity status:

   - Off - No power to instrument
   - Green - Network connection is good
   - Red - Network connection has fault or not connected

3. **Probe Compensation connectors**:



The Probe Compensation connections provide a ground connector and 1 kHz square wave source for adjusting the high-frequency response of a passive probe (probe compensation). The oscilloscope uses this signal to automatically compensate supported probes. See *Compensate the TPP0500B or TPP1000 probes* on page 59.

You can also use the ground connector to attach an anti-static wrist strap, to reduce the chance of electrostatic damage (ESD) while you handle or probe the DUT.

4. **USB Host ports**:

USB ports are located at the lower right corner of the front panel. Connect USB flash drives to which you can save or recall data (such as instrument firmware updates, waveforms, settings, and screen captures), or connect peripheral devices such as a mouse or keyboard.

5. **Aux Trig trigger input connector**:

An SMA electrical cable connector to which you can connect an external trigger input signal.

6. **Acq** status LED:

Shows the instrument trigger/acquisition status:

- Green - Triggered
- Yellow - Armed but not yet triggered
- Red - Acquisition stopped

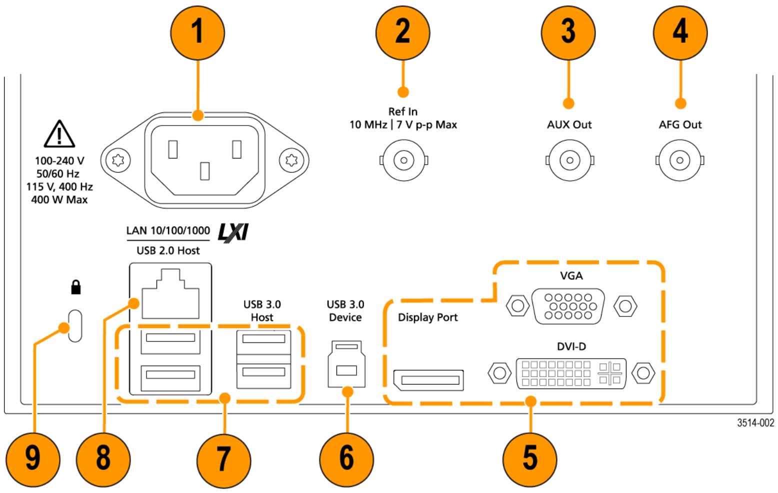
7. **Power On/Standby button**:

Powers the instrument on and off. The power button color indicates instrument power states:

- No light - No AC power applied
- Yellow - Standby mode
- Blue - Powered on

Rear panel connections

The rear panel connections supply power to the oscilloscope and provide connectors for network, USB devices, video, reference signals, and the AFG output.

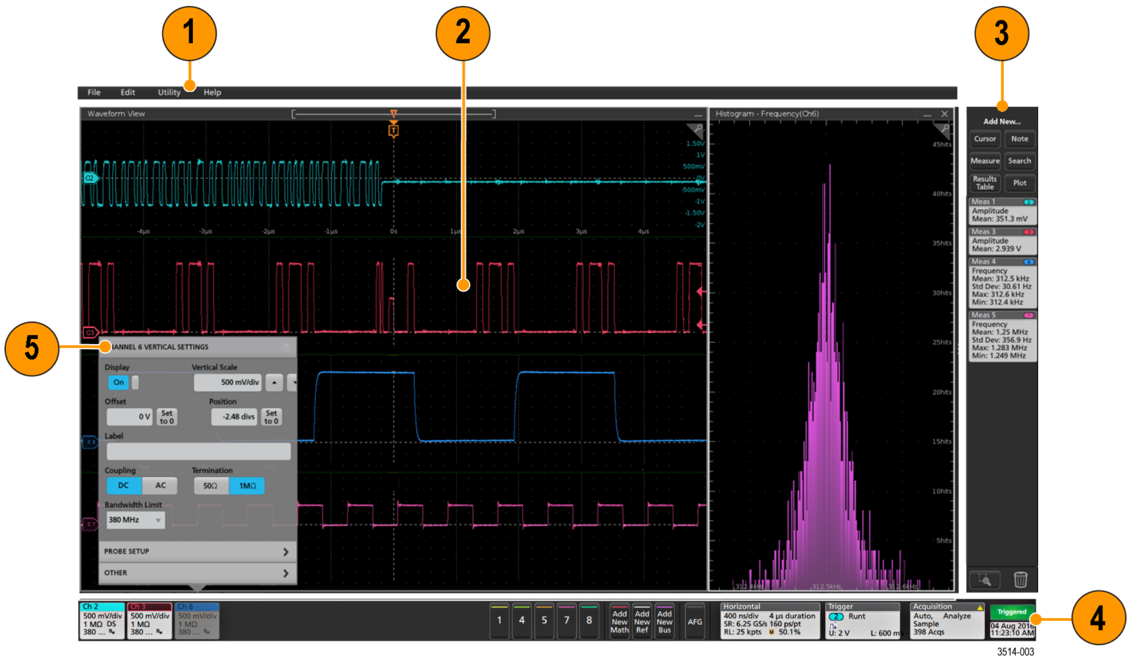


1. **Power cord** connector. Use only the power cord specified for this product and certified for the country of use.
2. **Ref In** lets you connect a high-precision 10 MHz reference signal to the oscilloscope for more accurate measurements.
3. **AUX Out** generates a signal transition on a trigger event, outputs a 10 MHz reference signal, or outputs a synchronization signal from the AFG.
4. **AFG Out** is the signal output for the Arbitrary Function Generator (AFG).
5. **Video outputs** (Display Port, VGA, and DVI-D) let you connect an external monitor or projector to show the oscilloscope screen.
6. **USB Device** port lets you connect to a PC to remotely control the oscilloscope using USBTMC protocol.
7. **USB Host** ports let you connect a USB memory device, keyboard, or mouse.
8. **LAN** connector (RJ-45) connects the oscilloscope to a 10/100/1000 Base-T local area network.
9. **Security** lock connector lets you use a standard PC/laptop lock cable to secure the oscilloscope to a work bench or other location.

The user interface screen

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

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.



1. The **Menu bar** provides menus for typical operations including:
   - Saving, loading, and accessing files
   - Undoing or redoing an action
   - Setting oscilloscope display and measurement preferences
   - Configuring network access
   - Running self tests
   - Erasing measurement and settings memory (TekSecure™)
   - Loading option licenses
   - Opening a Help viewer.
2. The **Waveform View** area displays analog, digital, math, reference, bus, and trend waveforms. The waveforms include waveform handles (identifiers), individual vertical graticule scale labels, and trigger position and level(s) indicators. You can set the Waveform View to stack each waveform vertically in separate graticules, called 'slices' (the default mode, as shown in the previous image), or overlay all the waveforms on the screen (traditional waveform view). See *The user interface elements* on page 42.

   You can also add Histogram, Spectral, Eye, and 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.
3. The **Results Bar** contains controls for displaying cursors, adding notes, plots, and result tables to the screen, and add measurements to the Results bar. 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 **Results Table** button adds a Measurement or Bus Results table to the screen. The Measurement Results table displays all measurements present in the Results bar. The Bus Results table displays bus decode information for

displayed bus waveforms. Each table is contained within its own view window, which can be moved within the display area.

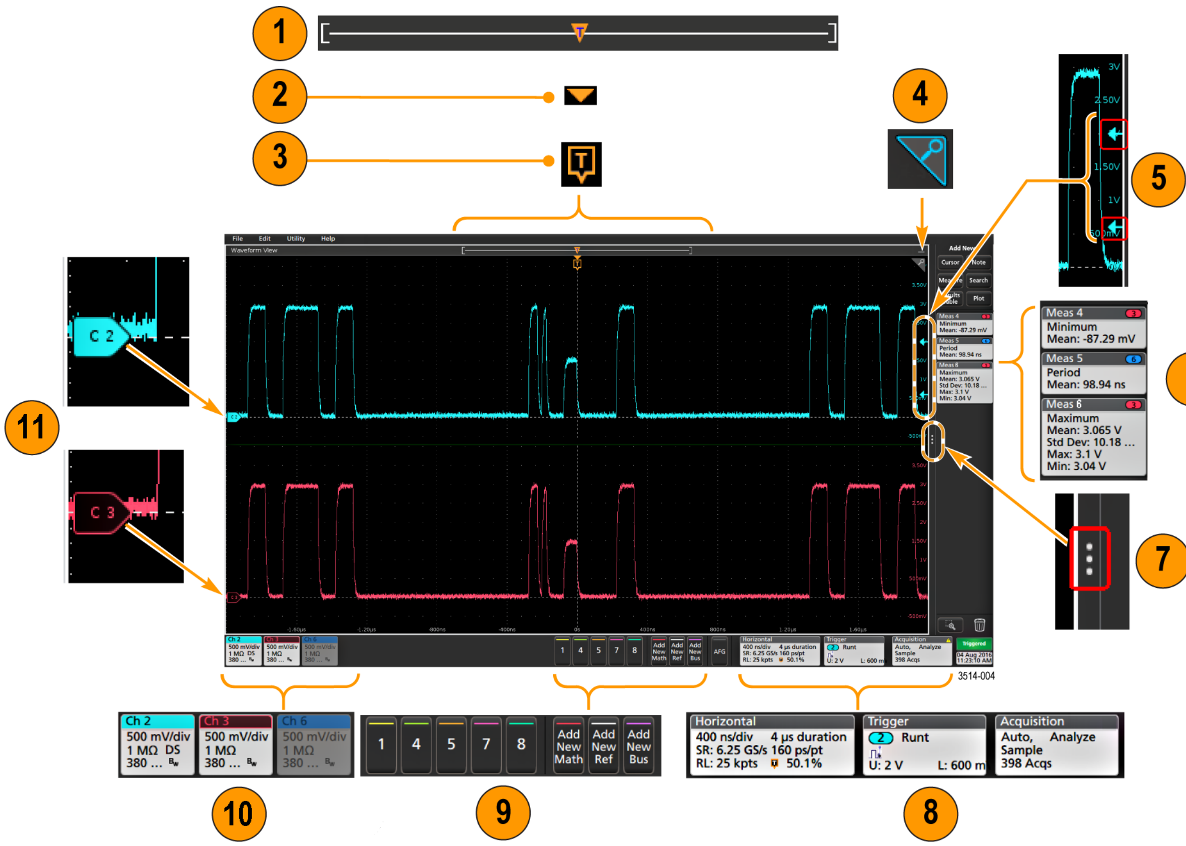
- The **Note** button adds a note object to the selected view. Double-tap the note text to open a configuration menu to change the text and font characteristics. Drag the note to any location on the view. Notes cannot be added to a **Results Table** view.
- The **Search** button lets you detect and mark a waveform where specified events occur. Tap **Search** to open a Search configuration menu and set the search conditions for analog and digital channels. You can add any number of searches to the same waveform or to different waveforms. Search badges are added to the **Results** bar.
- The **Plot** button adds an XY, XYZ, or Eye Diagram plot to the display. These plots are contained within their own window and can be moved within the overall display area.
- The **Measurement** and **Search** badges show measurement and search results. and are displayed in the middle area of the **Results** bar. See *Badges* on page 45. See *Add a measurement* on page 78. See *Add a Search* on page 87.
- The **Draw-a-Box** button at the bottom of the Results Bar lets you draw a box on the screen to zoom in on an area of interest.
- The **Trash Can** icon lets you drag Channel, Waveform, Measurement, and Search badges to the **Trash Can** to delete them.

4. The **Settings Bar** displays System badges for setting Horizontal, Trigger, Acquisition, 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; and Channel and Waveform badges that let you configure the individual waveform parameters. Tap a channel or waveform button to add it to the screen and display a badge. Double-tap any type badge to open its configuration menu. See *Badges* on page 45.
5. **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. See *Configuration menus* on page 50.

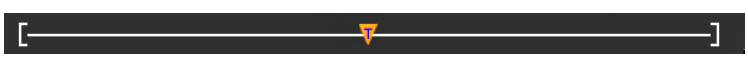
The user interface elements

Each area of the user interface has a specific function that helps manage information or controls. This topic shows and describes the key user interface elements.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.



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 record length.

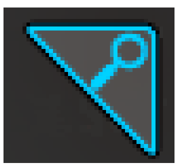
If cursors are active on a waveform, the Waveform record view shows the relative cursor positions as small vertical dashed lines.

When in Zoom mode, the Waveform Record View is replaced with the Zoom Overview. See *The Zoom user interface elements* on page 52.

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 (in upper right corner of Waveform and Plot views) toggles zoom on and off. The front panel **Zoom** button and knobs also turn on zoom mode and change the position and horizontal size of the Zoom Box.



5. The **Trigger Level Indicator** icon(s) shows the trigger level on the trigger source waveform. Some trigger types require two trigger levels.
6. **Measurement** and **Search** badges show measurement and search results. See *Badges* on page 45. See *Add a measurement* on page 78.
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, Acquisition, Run/Stop status, and Date/Time). See *Badges* on page 45.
9. The **Inactive Channel** buttons add channel waveforms to the Waveform view and add an associated Channel badge to the Settings bar.

   The **Add New** buttons add Math, Reference, and Bus waveforms to the Waveform view, and add an associated Waveform badge to the Settings bar. You can add any number of Math, Reference, and Bus waveforms, limited only by system memory.

   The optional **AFG** button opens the AFG configuration menu to set and enable the AFG output. This button is only present if the AFG option is installed.

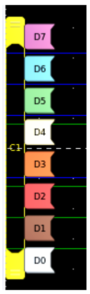
   The optional **DVM** button lets you use an analog probe to take DC, AC RMS, or DC+AC RMS voltage measurements on your DUT. Tap the button to add a **DVM** badge to the Results Bar and open a configuration menu. The DVM option also enables a trigger frequency counter, accessible from the **Mode & Holdoff** panel in the **Trigger** badge menu. This button is only present if the DVM option is installed.
10. **Channel** and **Waveform** badges (Math, Reference, Bus) show active channel and waveform settings and their status (selected, unselected, inactive). Double-tap a badge to open its associated configuration menu. See *Badges* on page 45. See *Configuration menus* on page 50.

If you add more Channel or Waveform badges than can fit in the waveform badge area, use the scroll buttons at each end of the waveform badge area to let you scroll through all badges.

11. The **Waveform Handles** identify the channel or source waveform (C1-C8 for channels, M*x* for Math waveforms, R*x* for Reference waveforms, B*x* for bus waveforms). The waveform handles are at the zero-volt level of the waveform. 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.

    For digital channels, the waveform handle shows the channel number, with the individual digital signal handles marked D0–D7. The colors follow a color code similar to that used on resistors. The D0 indicator is white, the D1 indicator is brown, the D2 indicator is red, and so on.



    Double-tapping a digital waveform handle opens the digital channel configuration menu.

    Dragging a digital signal handle over another handle swaps those two signals on the waveform.

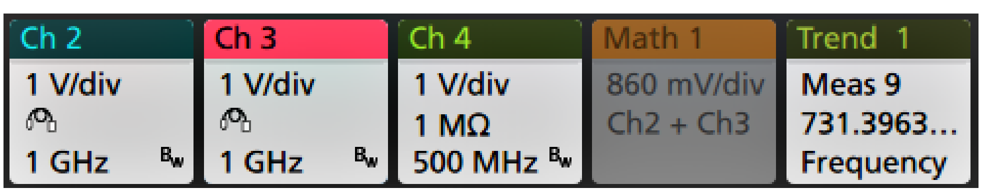
Badges

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, Search, and System.

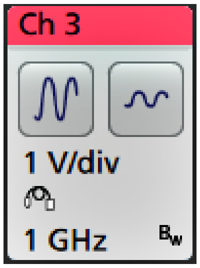
To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

Channel and Waveform badges

Channel and **Waveform** badges are located in the Settings Bar, located along the bottom left of the screen. These badges show settings for each displayed channel or waveform. Each waveform (channel, math, reference, bus, and trend) has its own badge. 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.

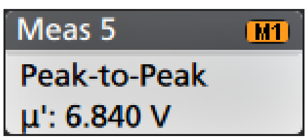


Channel badges are listed in the channel order. You cannot move **Channel** badges other than to drag them into the **Trash Can** icon to turn them off.

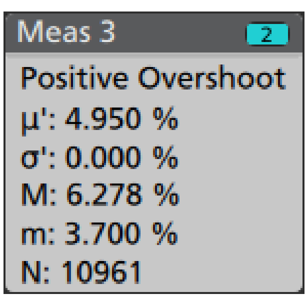
Waveform badges (Math, Ref, Bus, Trend) are listed in the order created, and are grouped together by type. Deleting a **Waveform** badge does not change the order or names of the remaining badges. You cannot move **Waveform** badges other than to drag them to the **Trash Can** icon to delete them.

Measurement badges

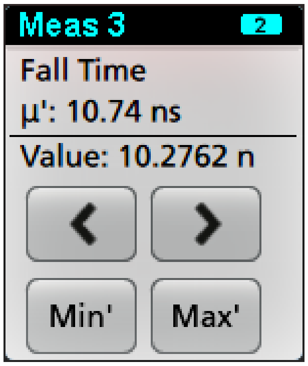
Measurement badges are located in the Results Bar. They show measurements or search results. The badge title also shows the measurement source or sources. To add a Measurement badge, tap the **Add New Measurement** button and select a measurement. Double-tap a Measurement badge to open its configuration menu to change or refine settings. The default measurement badge readout shows the measurement's mean (μ) value.



To add statistical readouts to individual measurement badges, double-tap a measurement badge to open its configuration menu and select **Show Statistics in Badge**.



Some Measurement badges also have Navigation buttons, shown by single-tapping the badge.



The **<** (Previous) and **>** (Next) buttons center the waveform in the display at the position of the previous or next measurement point in the record (for measurements that take more than one measurement per acquisition).

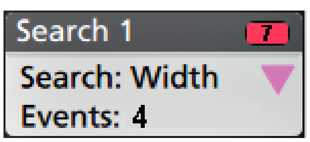
The **Min'** and **Max'** navigation buttons center the waveform in the display at the minimum or maximum value for that measurement in the current acquisition.

The prime symbol (') shown on measurement readings and Min/Max buttons indicates that the value shown (or moved to in the case of **Min**/**Max** and waveforms) is from the current acquisition. Lack of a prime symbol means the value is from all acquisitions.

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. You cannot move **Measurement** badges other than to drag them into the **Trash Can** icon to delete them.

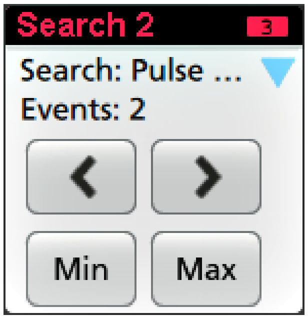
Search badges

Search badges are also located in the Results Bar, below the Measurement badges. A search badge defines the search source, type, and the number of search criteria events (occurrences) in the current acquisition, and marks the waveform where those events occur with small down-pointing triangles along the top of the waveform graticule. Double-tap a search badge to open its configuration menu to change or refine search settings.



Search badges are created by tapping the **Add New... Search** button. Use the displayed configuration menu to set the search criteria.

Search badges have **<** (Previous) and **>** (Next) Navigation buttons that open the Zoom mode and center the waveform in the display at the position of the previous or next search mark in the waveform record. Search badge Navigation buttons are only usable when the oscilloscope is in single acquisition mode. Single-tap a badge to close the Navigation buttons.



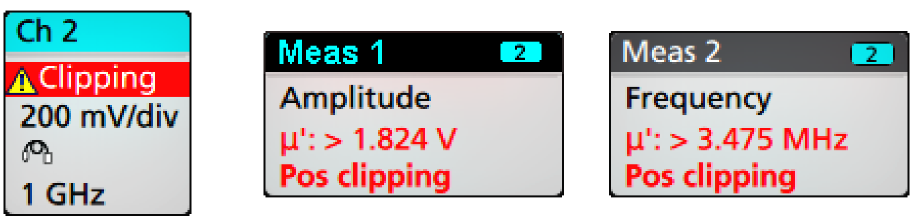
Some searches also provide **Min** and **Max** navigation buttons that open the Zoom mode and center the waveform in the display at the minimum or maximum value for that search event in the current acquisition.

Search badges are listed in the order created. Deleting a **Search** badge does not change the order or names of the remaining badges. You cannot move **Search** badges other than to drag them into the **Trash Can** icon to delete them.

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).

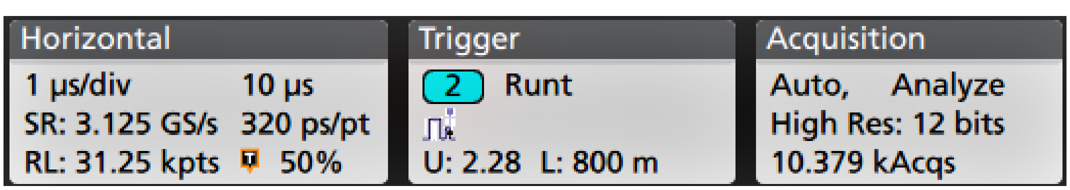


To close the clipping message, change the vertical scale to show the entire waveform, disconnect the probe tip from the excessive voltage source, and check that you are probing the correct signal using the correct probe.

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

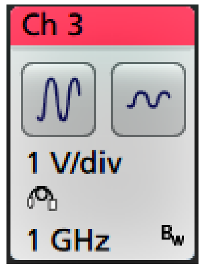
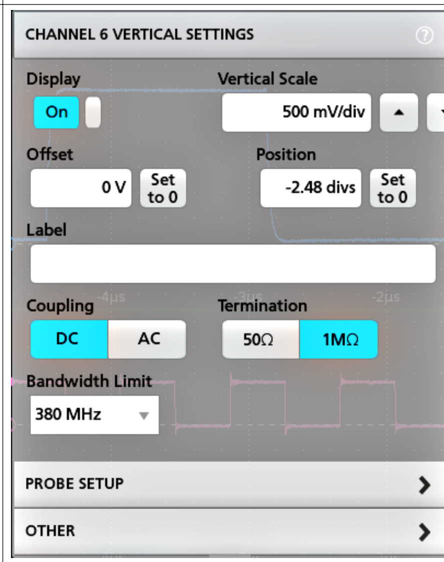
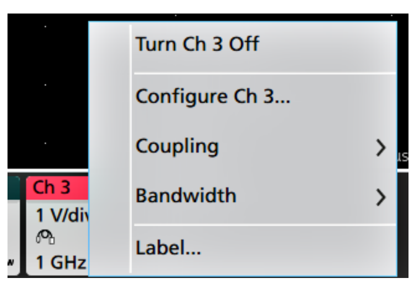
System badges (in the Settings bar) display the main Horizontal, Trigger, and Acquisition settings. You cannot delete System badges.



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

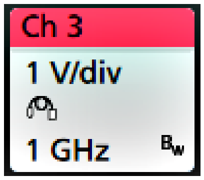
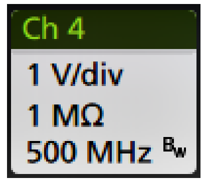
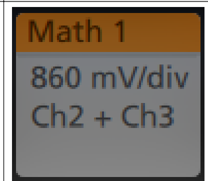
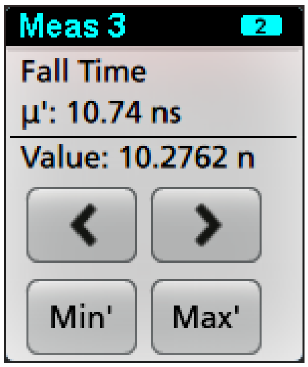
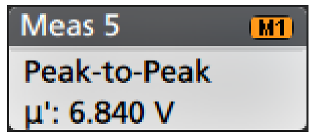
The Horizontal badge also has Scale buttons, shown by single-tapping the badge. Use the Horizontal Scale buttons to increase or decrease the horizontal time setting.

Common badge actions

Action	Result	Example
Single tap	Immediate access controls (Scale, Navigation).	
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.	

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	Turned off or in use [1]
Channel or Waveform			
Measurement			N/A

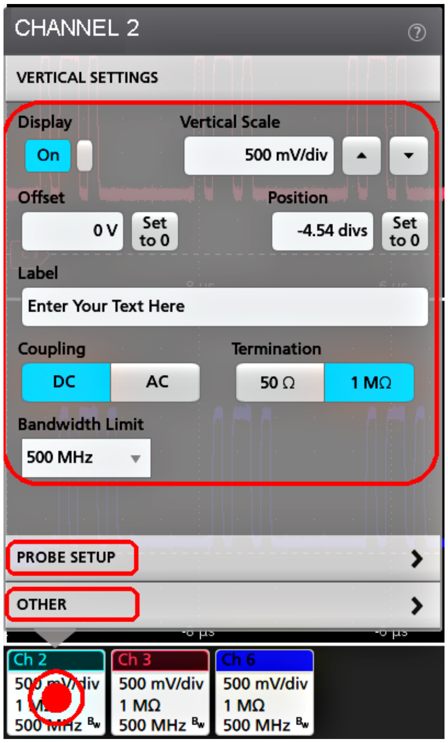
Configuration menus

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

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

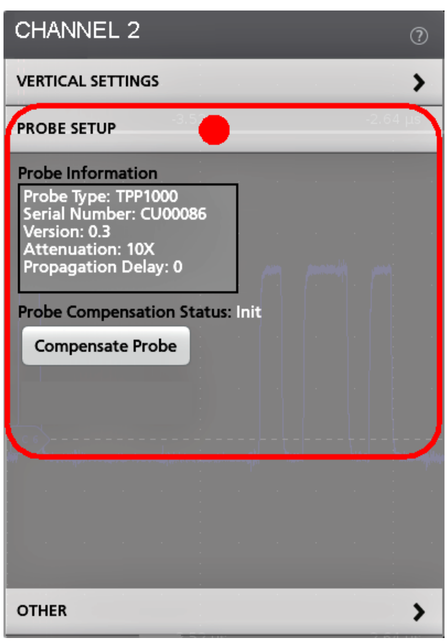
Double-tap an item (badge, Waveform or Plot view, cursor readouts, note text, and so on) to open its configuration menu. For example, double-tap a Channel badge in the Settings bar to open its configuration menu.

[1] A dimmed Channel badge means the screen waveform is turned off (but not deleted). A dimmed Waveform badge means that the waveform display is turned off, or it is being used as a source by a measurement and cannot be deleted until the measurement is deleted.



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

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.



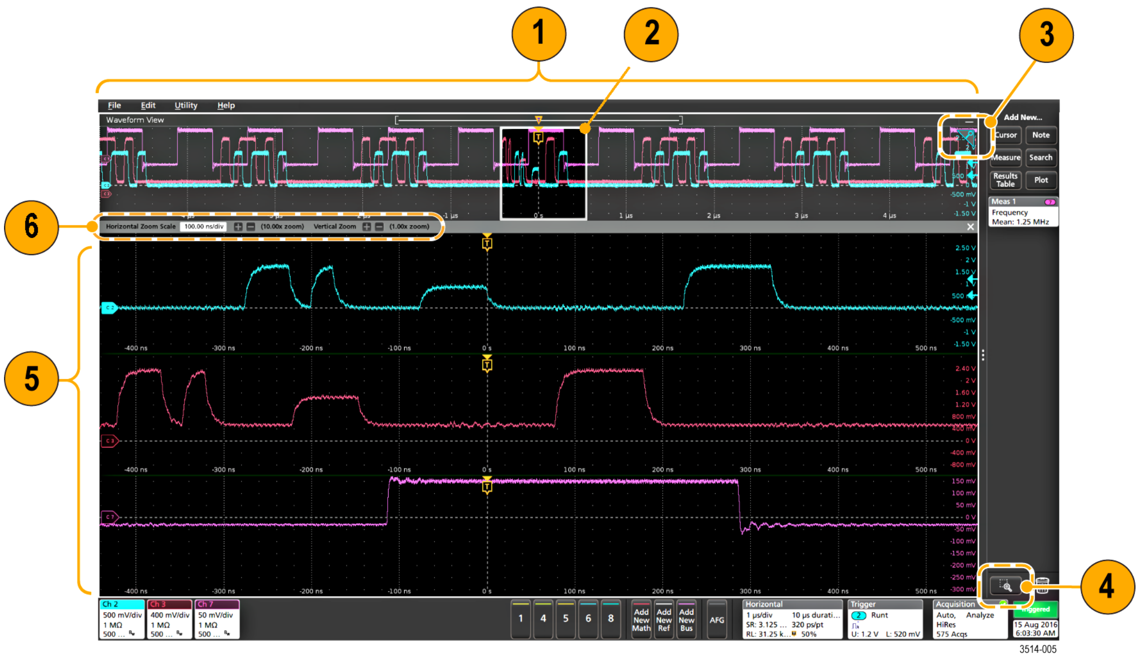
Tap anywhere outside a configuration menu to close it.

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

The Zoom user interface elements

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

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.



1. The **Zoom Overview** shows the entire waveform record. All waveforms are shown in Overlay mode in the Zoom Overview area.

   NOTE. *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. You can touch and drag the box to move the area to view. You can also use the zoom **Pan** knob to move the Zoom Box left or right.

   NOTE. *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. The **Draw-a-Box** button lets you quickly draw a box around an area of interest in the Waveform or Zoom Overview on which to zoom. Drawing a box immediately puts the oscilloscope into zoom mode. The button is located at the bottom of the Results bar. Tap the button, then touch and drag on the waveform to draw a box on the area of interest.

   To keep the Draw-a-Box button enabled, double-tap the Draw-a-Box button. Keeping the Draw-a-Box function enabled lets you update your Zoom display by drawing new boxes in either the Waveform Slice or Zoom Overview window.

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.

   NOTE. *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.

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.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

The oscilloscope has a user interface tutorial. Tap **Help > User Interface Tutorial** to quickly learn the fundamental touch operations.

Table 1: Common touchscreen UI tasks (with mouse equivalents)

Task	Touchscreen UI action	Mouse action
Add a channel, math, reference, or bus waveform to the screen.	Tap an inactive channel button, Add New Math, Add New Reference, or Add New Bus button.	Click an inactive channel button, Add New Math, Add New Reference, or Add New Bus button.
Select a channel, math, reference, or bus waveform to make it active	Stacked or Overlay mode: Tap the Channel or Waveform badge. Stacked mode: Tap the channel, math, reference, or bus waveform slice or handle. Overlay mode: Tap the channel or waveform handle.	Stacked or Overlay mode: Left-click the Channel or Waveform badge. Stacked mode: Left-click the channel, math, reference, or bus waveform slice or handle. Overlay mode: Left-click the channel or waveform handle.
Display scale or navigation buttons on a badge (waveform, measurement [2], search, 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.

[2] Not all measurement or search badges display navigation buttons.

Task	Touchscreen UI action	Mouse action
Close a configuration menu [3].	Tap anywhere outside the menu or dialog.	Click anywhere outside the menu or dialog.
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.
Move a note [4].	Touch and hold on a note and quickly [5] start to drag, then move to new position.	Click and hold the right mouse button on the note and quickly start to drag, then move to the 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. Pinch or expand two fingertips on the waveform view, move them together or apart vertically or horizontally, lift from screen; repeat.	Left-click a channel, waveform, or Horizontal badge and click on the Scale buttons.
Increase or decrease the zoom area (while in Zoom mode)	Pinch or expand 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) or anywhere in the divider between the Waveform View and the Results Bar.	Click the Results Bar Handle (three vertical dots) or anywhere in the divider between the Waveform View and the Results Bar. Click and drag the Results Bar divider.

Accessing application help

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

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

[3] Some dialog boxes will not close until you click an OK, Close, or other button in the dialog.
[4] Notes are screen objects and are not associated with any particular waveform channel or slice.
[5] Start to move the note as soon as selected (highlighted), otherwise the UI opens the right-click menu.

Using context-sensitive help

To open help on a particular menu or item, tap the Help button (question mark symbol) in the title bar. The browser opens with content relevant to the menu or item.

Navigating the online help

Select **Help > Help**. The Help Browser is similar in look and feel to PC-based Help tools. From the help browser, select one of the following tabs:

- **Contents Tab**. Click any entry to display information on the subject.
- **Index Tab**. Double-click an entry to display information on the subject. Or, enter a keyword you are looking for (the list scrolls to that topic). Click Display to open the topic.
- **Search Tab**. Type in the keyword you are looking for; then click List Topics. Every topic that contains the keyword will be displayed. Select a topic, and then click Display to open the topic.
- **Bookmarks Tab**. To save the current topic to a list, click the Bookmarks tab; then right-click and select Add. Double-click to return to a topic at any time.

Other help features

- Zoom on help text. Use the magnifying glass icons to zoom in or out of the help text.

Configure the instrument

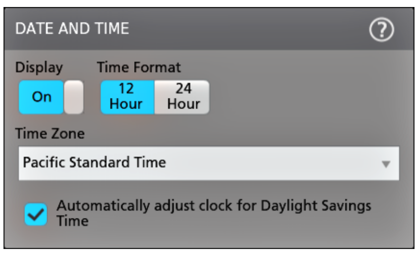
Set the time zone and clock readout format

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).

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

To remotely set this control or run this task on an MSO58LP, see the *5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx) for the correct command or commands to use.

1. Double-tap the Date/Time badge (bottom-right of screen) to open the configuration menu.



2. Select your local time zone and the time format (12 or 24 hour). Settings take place immediately.
3. To turn off showing the date and time on the screen, tap the **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**.
4. 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.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

To remotely set this control or run this task on an MSO58LP, see the *5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx) for the correct command or commands to use.

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.
5. Push the **Autoset** button. You should see a square wave in the display (approximately 5 V_{P-P}).
6. Tap the **Add New...Measure** button.
7. Tap the **Timing 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.

Download and install the latest firmware

Loading the latest firmware helps ensure that your oscilloscope is taking the most accurate measurements. You can check for the latest updates from network-connected oscilloscopes, or download the update file to a USB drive and install from there.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

Prerequisite: Save any important on-instrument files (waveforms, screen captures, oscilloscope setups, and so on) to a USB drive or network.

Update oscilloscope firmware from USB drive

Prerequisite: Determine the current version of firmware installed on the oscilloscope (**Help > About**)

1. Open up a Web browser on a PC and go to **www.tektronix.com/software**.
2. Enter **MSO5** in the search field and click **Search**.
3. Select **Software** in the **Filter by type** list.
4. If the listed available firmware version is newer than what is on your oscilloscope, select and download that file to your PC.
5. Follow the installation instructions that came with the downloaded firmware to create a firmware install file.
6. Copy the firmware install file to a USB drive.
7. Insert the USB drive into any oscilloscope USB Host port. The oscilloscope detects the USB drive with the firmware file and starts the installation process.
8. Follow instructions to install the firmware.

NOTE. *Do not power off the oscilloscope or remove the USB flash drive until the oscilloscope finishes installing the firmware. The oscilloscope displays a message when it is OK to turn off the oscilloscope.*

---

9. When the firmware install is finished, remove the USB drive and restart the oscilloscope.

To confirm the firmware installation:

1. Tap **Help > About** in the Menu bar.
2. Verify that the firmware version number listed on the screen is the same version that you downloaded.

Run Signal Path Compensation (SPC)

Run SPC at regular intervals for best measurement accuracy. You should run SPC whenever the ambient (room) temperature has changed by more than 5 °C (41 °F), or once a week if you use vertical scale settings of 5 mV per division or less.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

To remotely set this control or run this task on an MSO58LP, see the *5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx) for the correct command or commands to use.

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 oscilloscope not meeting warranted performance levels at low volts per division settings.

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

1. Power on and warm up the oscilloscope 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 several minutes per channel to run, so wait until the SPC Status message changes to Pass before reconnecting probes and using the oscilloscope.

⚠ ***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.
5. 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 TPP0500B or TPP1000 probes

Probe compensation adjusts the high frequency response of a probe for best waveform capture and measurement accuracy. The oscilloscope can automatically test and store compensation values for an unlimited number of probe/channel combinations.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

To remotely set this control or run this task on an MSO58LP, see the *5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx) for the correct command or commands to use.

The oscilloscope stores the compensation values for each probe/channel combination, and automatically recalls the compensation values when you plug in the probe. Probe compensation status is shown in the Probe Setup panel of the Channel configuration menu.

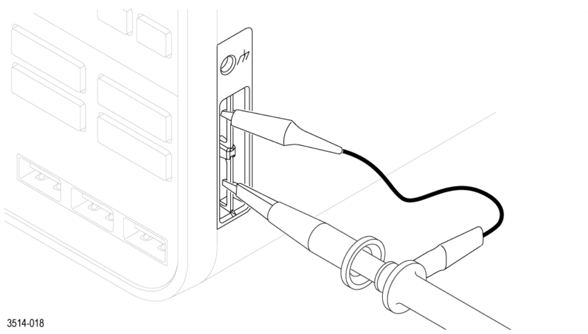
- If the Probe Compensation Status field displays **Pass**, the probe is compensated and ready for use.
- If the Probe Compensation Status field displays **Default**, the attached probe has not been compensated and needs to have this probe compensation procedure run.
- If the Probe Compensation Status field displays **Fail**, the attached probe has failed the probe compensation procedure. Reconnect the probe and run probe compensation again.
- If there is no probe compensation status field shown in the panel, the oscilloscope cannot store compensation values for that probe. See the oscilloscope Help for how to manually compensate passive probes not supported by the probe compensation function.

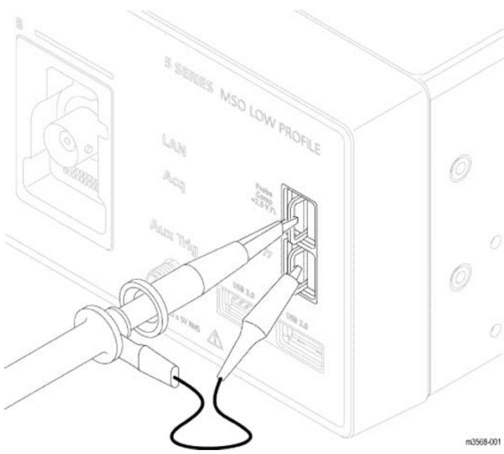
Use this procedure to compensate a TPP0500B, TPP1000, or other supported TPP-family probe that shows a Default status when connected to the oscilloscope.

***NOTE.** A **Default Setup** does not delete probe compensation values. A factory calibration deletes all stored probe compensation values.*

Prerequisite: The oscilloscope must be powered on for at least 20 minutes before compensating a probe.

1. Connect a supported probe to an input channel. The probes shipped with the oscilloscope support this probe compensation process.
2. Connect the probe tip and ground lead of the probe to the PROBE COMP terminals on the lower right of the oscilloscope (near the Default Setup and Autoset front panel buttons).





Connect the probe tip to the 1 kHz source, and the ground clip to the ground. For best results, remove any probe tip accessories and hold the probe tip directly onto the 1 kHz connector.

***NOTE.** Connect only one probe at a time to the PROBE COMP terminals.*

---

3. Turn off all channels.
4. Turn on the channel to which the probe is connected.
5. Push the front-panel **Autoset** button. The screen displays a square wave.
6. Double-tap the badge of the channel you want to compensate.
7. Tap the **Probe Setup** panel.

If the Probe Compensation Status says **Pass**, the probe is already compensated for this channel. You can move the probe to another channel and start again from step 1, or connect a different probe to this channel and start from step 1.

If the Probe Compensation Status says **Default**, continue with this procedure.

8. Tap **Compensate Probe** to open the **Probe Compensation** dialog.
9. Tap **Compensate Probe** to run the probe compensation.
10. The probe compensation is finished when the Probe Compensation Status displays **Pass**. Disconnect the probe tip and ground from the PROBE COMP terminals.
11. Repeat these steps for each supported passive probe that you want to compensate for this channel.
12. Repeat these steps to compensate supported probes on other channels of the oscilloscope.

NOTE. *For most accurate measurements, open the* ***Probe Setup*** *panel and verify the Probe Compensation Status is* ***Pass*** *whenever you attach a probe to a channel.*

Compensate passive probes

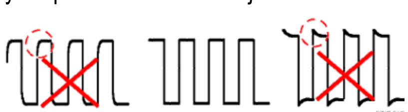
Probe compensation adjusts the high frequency response of a probe for best waveform capture and measurement accuracy. Use this procedure to adjust probe compensation for probes with a manual adjustment.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

To remotely set this control or run this task on an MSO58LP, see the *5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx) for the correct command or commands to use.

A passive probe is only adjusted for one channel at a time. If you move a passive probe to another channel, you must compensate that probe to that channel.

1. Connect the probe to the channel where you want to use it to take measurements. Remove all other probes.
2. Turn on the channel to which the probe is connected. Turn off all other channels.
3. Attach the probe tip and reference lead to the probe compensation connectors.
4. Push the **Autoset** button to display a square wave.
5. Adjust the vertical **Scale** and **Position** knobs to display as large a waveform as possible.
6. Use the adjustment tool provided with the probe to adjust the probe until the square wave has as flat a top as possible. See your probe manual for adjustment location and instructions.



Connect to a network (LAN)

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

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

To remotely set this control or run this task on an MSO58LP, see the *5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx) for the correct command or commands to use.

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 oscilloscope 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, 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 connections 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 rear of the instrument.
5. If using the USB connection to remotely control the oscilloscope using GPIB commands, set the **GPIB Talk/Listen Address** for your configuration (0 - 30).

Remote access from a Web browser

You can remotely control your network-connected oscilloscope from a Web browser. Enter the IP address into your Web browser to display the oscilloscope screen and use a mouse to access all screen-based controls and items on the oscilloscope.

Prerequisites:

- The oscilloscope must be connected to, and accessible from, the network to which the PC is connected. See *Connect to a network (LAN)* on page 62
- The IP address of the oscilloscope that you want to access. To determine an oscilloscope's IP address, select **Utility > IO** in the oscilloscope menu bar and view the network settings in the **LAN** panel.

1. Open a Web browser on a PC connected to the same network as the oscilloscope.
2. Enter just the oscilloscope IP address on the URL line of the browser and press **Enter**. For example: 135.62.88.157.

The browser searches for and opens the Web page for the oscilloscope.

3. Select **Instrument Control (e*Scope®)**.

   The browser displays the instrument screen.

4. Use a mouse to select and interact with the oscilloscope controls shown in the Web browser. If your remote PC or laptop has a touch screen monitor, you can use that to access the oscilloscope controls.

Deskew analog input channels - quick visual method

Use the following procedure to visually align waveform edges to compensate for timing differences between probes.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

To remotely set this control or run this task on an MSO58LP, see the *5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx) for the correct command or commands to use.

Critical timing measurements on multiple channels require that all probes be adjusted, or deskewed, to compensate for signal timing differences between probes. This procedure uses displayed waveform edges to quickly minimize deskew between probes.

NOTE. *Once probes have been deskewed for a particular channel, you should only use the probes on the channels for which they were deskewed, when taking critical timing measurements.*

1. Connect all probes that you want to deskew.
2. Connect all probe tips and ground leads to the Probe Compensation connector (maximum of four channels at a time).
3. Turn on all the channels that you want to deskew.
4. Double-tap in the Waveform view and set **Waveform Mode** to **Overlay**.
5. Push the **Autoset** button.
6. Adjust the vertical **Scale** and **Position** controls for each channel so that the signals overlap and are centered on the display.
7. Adjust the horizontal **Scale** so that the differences in the channel delays are clearly visible.
8. Determine the channel you want to use as your reference.
9. Double-tap the Channel badge of a channel other than the reference channel and tap the **Other** panel.
10. Tap the **Deskew** field and use the multipurpose knob to align this channel with the reference channel waveform such that the waveforms cross the trigger point at the same time. For fine adjust, double-tap the Deskew field to open a number pad.
11. Repeat steps 9 and 10 for each additional channel you want to deskew.

Deskew analog input channels - measurement method

Use the following procedure to more accurately minimize timing differences between probes.

To access the user interface on the MSO58LP, connect a monitor to a video port on the rear of the instrument, and connect a mouse to any USB Host port. You do not need to connect a mouse if your remote monitor is touch-capable. You can also remotely access the user interface of a network-connected instrument by entering the instrument's IP address in a web browser.

To remotely set this control or run this task on an MSO58LP, see the *5 Series MSO MSO54, MSO56, MSO58, MSO58LP Programmer Manual* (Tektronix part number 077-1305-xx) for the correct command or commands to use.

Critical timing measurements on multiple channels require that all probes be adjusted, or deskewed, to compensate for signal timing differences between probes. This procedure uses a Delay measurement to adjust a probe's deskew setting.

NOTE. *Once probes have been deskewed for a particular channel, you should only use the probes on the channels for which they were deskewed, when taking critical timing measurements.*

1. Connect all probes that you want to deskew.
2. Connect all probe tips and ground leads to the Probe Compensation connector (maximum of four channels at a time).
3. Turn on all the channels that you want to deskew.
4. Push the **Autoset** button.
5. Change the vertical Scale of all active channels to **500mV/div** and adjust the vertical Position so that the waveforms are centered in their respective slices.
6. Determine the channel you want to use as your reference.
7. Tap the **Add New...Measure** button and tap the **Timing Measurements** panel.
8. Select the **Delay** measurement and set your chosen reference channel as **Source 1** and your channel being deskewed as **Source 2**, then tap the **Add** button.
9. Double-tap the Channel badge of the channel being deskewed (Source 2)and tap the **Other** panel
10. Tap the **Deskew** field and use the multipurpose knob to align this channel with the reference waveform such that the measured delay between channels becomes a minimum. For fine adjust, double-tap the Deskew field to open a number pad.
11. Double-tap the **Delay** measurement badge and set the Source 2 channel to the next channel to deskew.
12. Repeat steps 9 through 11 for each additional channel you want to deskew.

Connect a keyboard or mouse

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

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

1. Remove and reinsert the USB cable or dongle in the same port.
2. Insert the USB cable or dongle into a different USB port.

Connect an external monitor or projector

Use the video outputs to send the instrument display to a projector or to a flat-panel LCD monitor.

1. Power on the oscilloscope.
2. Connect the appropriate video cable to the projector or monitor. Connect the other end to the correct DVI, Display Port, or VGA connector on the oscilloscope.
3. Power on the projector or monitor.
4. Follow the projector or monitor instructions to set up and adjust the image.

Prevent ESD 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 TekConnect adapters. The instrument provides a front panel connection to attach a wrist strap.
- 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. Click **here** for details on setting up the Vertical input controls.

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

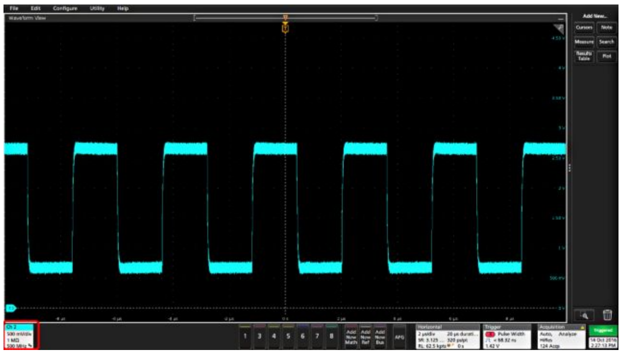
3. Tap the channel button to add the channel waveform to the waveform view and add a channel badge to the setting bar. A channel button lights when its channel is on.
4. Double-tap the channel badge to open the channel Vertical Settings menu. To change the input coupling, select the appropriate coupling 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 channels 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 knob 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 and sample rate (when in Manual mode).
9. You may attempt to stabilize the display by pressing the trigger Level control 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.

Quickly display a waveform (Autoset)

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 the lowest-numbered channel. The signal can be analog or digital.
2. Connect any other associated signal(s) to available channel input(s).

3. Add the channel waveforms to the Waveform view. *Add a channel waveform to the display* on page 72
4. Push the front-panel **Autoset** button. When using the Stacked Display mode, the instrument analyzes the signal characteristics of the lowest-numbered displayed channel (analog or digital) and adjusts the horizontal, vertical, and trigger settings accordingly to display a triggered waveform for that channel. Vertical scale is adjusted for all other active waveforms to maximize ADC utilization.



When using the Overlay Display mode, the instrument adjusts the horizontal and trigger settings of the lowest-numbered displayed channel to display a triggered waveform for that channel, and adjusts the vertical settings and position of all active waveforms such that they are uniformly spaced on screen.

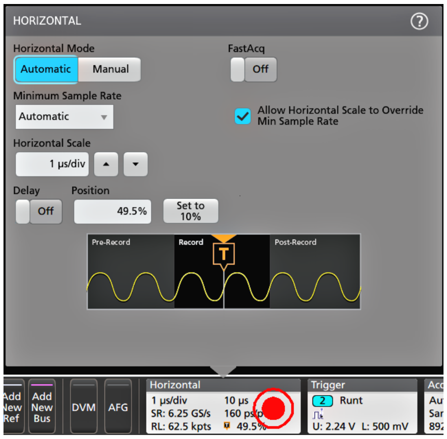
Autoset guidelines:

- Autoset displays two or three 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 view whether it has a signal or not.
- For digital signals, the oscilloscope analyzes and triggers on the lowest (LSB) digital channel with an active signal.
- 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, minimum sample rate, 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.



2. Use the menu selections to set horizontal parameters.

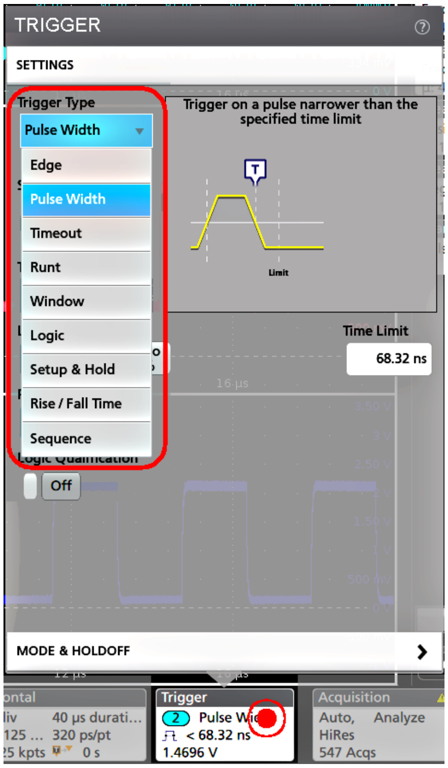
3. Tap the Help icon on the menu title for more information on these settings.

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.

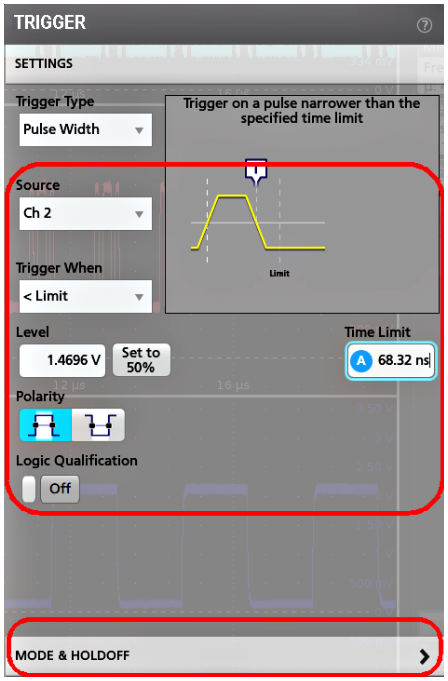


NOTE. *To trigger on a bus, you must first add the bus to the Waveform view. See* *Add a math, reference, or bus waveform on page 77*

NOTE. *Triggering on buses other than Parallel requires purchasing and installing serial trigger and analysis options. See the Tektronix Web site for available serial trigger and analysis options.*

---

3. Select the other fields and panels to refine the trigger conditions. The menu fields and trigger graphic update as you make changes to the trigger settings. Displayed fields depend on the selected trigger type. Selection changes take effect immediately.

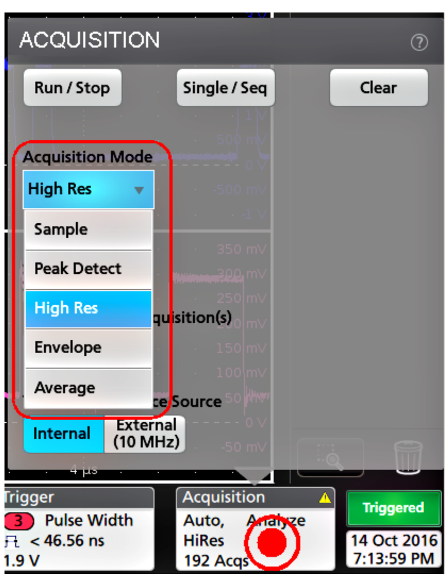


4. Tap the Help icon on the menu title for more information on these settings.
5. Tap outside the menu to close the menu.

Set the acquisition mode

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

1. Double-tap the **Acquisition** badge on the Settings bar to open the Acquisition configuration menu.
2. Select the acquisition method from the **Acquisition Mode** list. Set any other parameters associated with the selected acquisition type.

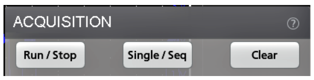


3. Tap the Help icon on the menu title for more information on these settings.
4. Tap outside the menu to close the menu.

Start and stop an acquisition

Acquisition controls the start and stop of waveform acquisition.

1. To start an acquisition, double-tap the Acquisition badge and tap **Run/Stop** in the Acquisition configuration menu. You can also push the **Run/Stop** button on the front panel.



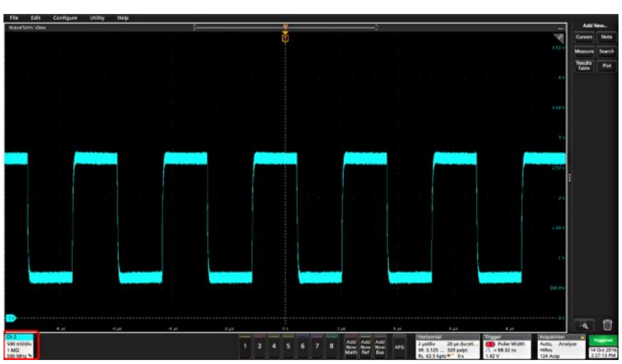
2. To stop an acquisition, tap **Run/Stop** again or push the **Run/Stop** button.
3. To take a single acquisition, double-tap the Acquisition badge and tap **Single/Seq** in the Acquisition configuration menu, or push the **Single/Seq** button on the front panel.
4. The color of the **Run/Stop** and **Single/Seq** buttons on the front panel indicate the acquisition status (green = acquiring, red = stopped). The Trigger
5. 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 (digital or analog). Channels are displayed from lowest-numbered channel at the top, to highest-numbered channel at the bottom of the view, regardless of the order they were added (in stacked mode).

4. Double-tap a channel badge to open that channel's configuration menu to check or change settings.

Configure channel and waveform settings

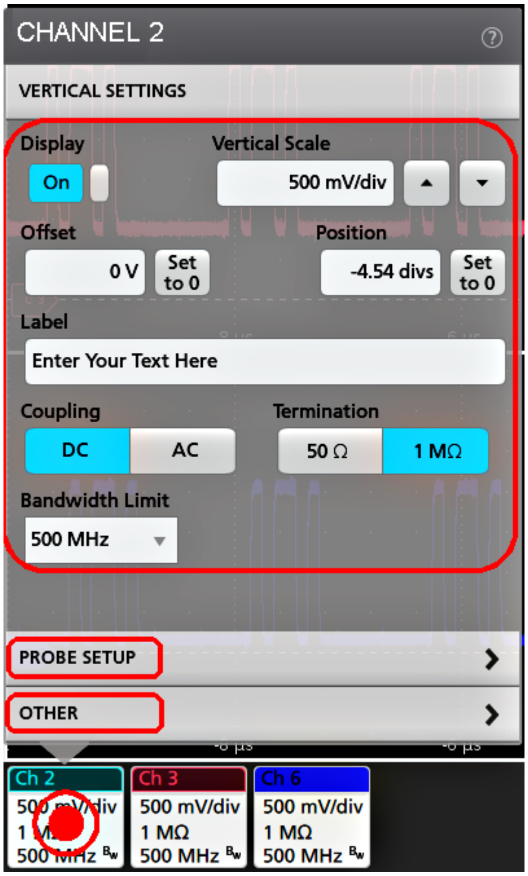
Use the channel and waveform configuration menus to set parameters such as vertical scale and offset, coupling, bandwidth, probe settings, deskew values, external attenuation values, and other settings.

Use this procedure to set up analog signal input.

Prerequisites:

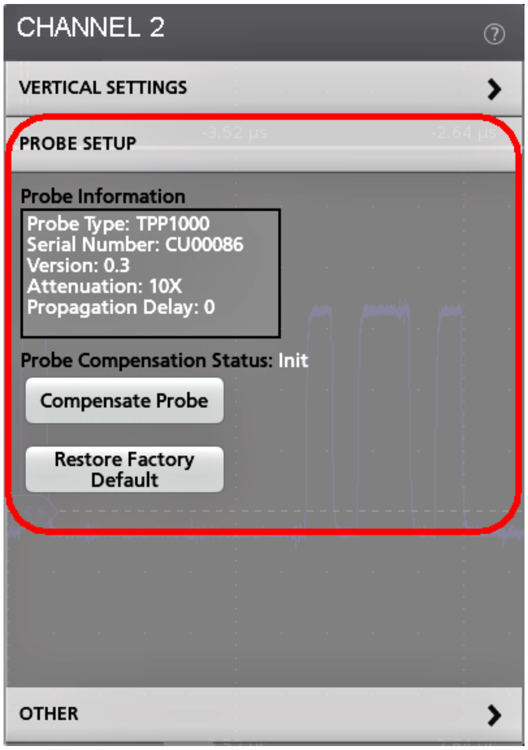
- An analog probe or cable is connected to the channel before setting up the channel.
- There is a channel or waveform badge in the Settings bar.

1. Double-tap a **Channel** or **Waveform** badge to open the configuration menu to the Vertical Settings panel. Displayed fields and controls can change depending on menu selections.



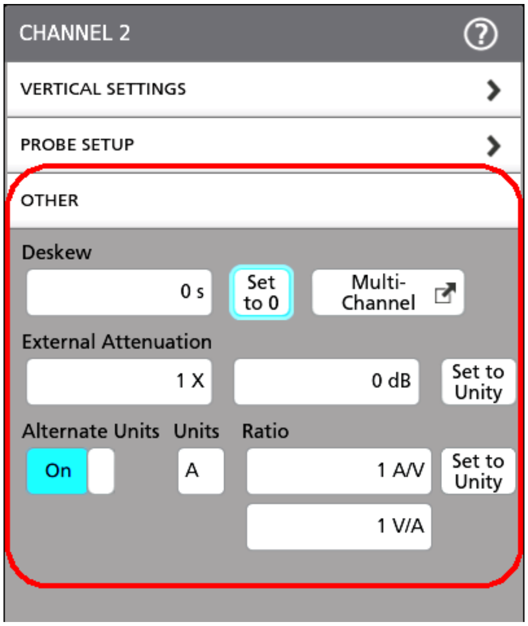
Field or control	Description
Display	Toggles display of the source waveform on or off.
Vertical Scale	Adjusts the vertical scale: Double-click the **Scale** control and use the virtual keypad to make the adjustment.
Offset	Adjusts the vertical offset: Double-click the **Offset** control and use the virtual keypad to make the adjustment.
Set to 0	Sets the vertical offset to 0.
Position	Adjusts the vertical position: Double-click the **Position** control and use the virtual keypad to make the adjustment.
Set to 0	Sets the vertical position to 0.
Label	Enter a label for the source waveform.
Coupling	Sets the coupling for the channel. If the termination is 50 Ω, only DC coupling is available.
Termination	Sets the termination for the channel. If DC coupling is selected and the attached probe supports it, 1 MΩ and 50 Ω options are available. Some probes automatically set the required termination.
Bandwidth Limit	Sets the bandwidth limit of the channel.

2. To set up your probe, tap the **PROBE SETUP** panel. The fields and controls in the Probe Setup panel vary with the type of probe connected to the channel. A TPP probe is shown in this example, for other probes, see the probe user documentation.



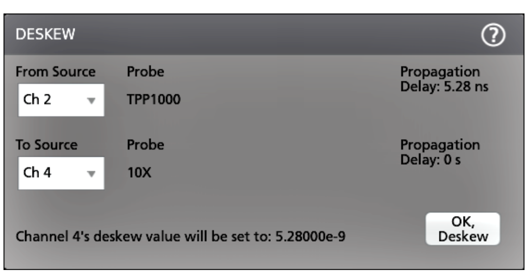
Field or control	Description
Probe Information	Displays the probe type, serial number, and attenuation, if it is available.
Probe Compensation Status	Displays the compensation status of the attached probe.
Compensate Probe	Starts the probe compensation procedure.
Restore Factory Default	Returns the compensation to the factory default settings.

3. To set up deskew, external attenuation, or alternate units, tap the **OTHER** setup panel.



Field or control	Description
Deskew	Allows setting the channel deskew value.
Set to 0	Sets the channel deskew to 0.
Multi-Channel	Brings up the Deskew configuration menu.
External Attenuation	Allows setting an external attenuation for the channel. As one field is edited, the other field changes to reflect the corresponding value.
Set to Unity	Sets left numeric input field to 1X and right input to 0.0 dB.
Alternate Units	Toggles alternate units on or off.
Units	Sets the ratio units. This field is disabled when the Alternate Units switch is off.
Ratio	Sets the desired ratio value. As one field is edited, the other field changes to reflect the corresponding value. This button is disabled when the Alternate Units switch is off.
Set to Unity	Sets the ratio to 1. This button is disabled when the Alternate Units switch is off.

4. When **Multi-Channel** in the **Other** panel is selected, the DESKEW configuration menu is displayed.



Field or control	Description
From Source	Selects the From Source to Deskew.
Probe	Displays the probe name or a drop-down list to select the probe connected to the From Source.
Propagation Delay	Displays the propagation delay of the probe shown in the Probe control.
To Source	Selects the To Source to Deskew.
Probe	Displays the probe name or a drop-down list to select the probe connected to the To Source.
Propagation Delay	Displays the propagation delay of the probe shown in the Probe control.
OK, Deskew	Uses the deskew values to adjust the horizontal delay between channels

5. Tap outside the menu to close the menu.

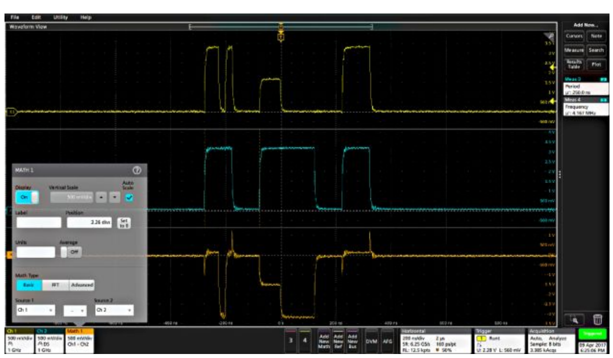
Add a math, reference, or bus waveform

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.

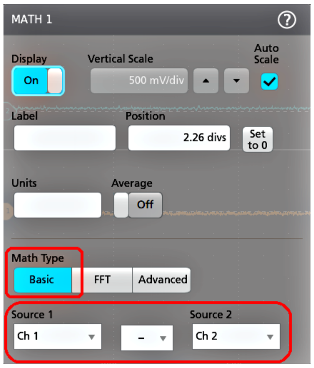
1. Tap the **Add New Math**, **Add New Ref**, or **Add New Bus** button in the Settings bar.

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, and set the math type to **Basic** math operation, and subtracting channel 2 from channel 1.

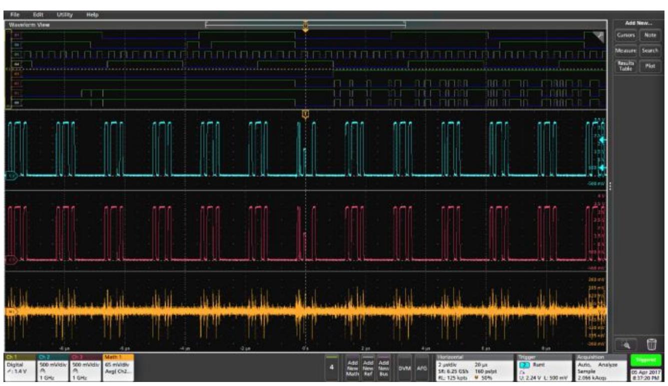


4. When adding a Reference waveform, the instrument presents a **Recall** configuration menu. Navigate to and select the reference waveform file (*.wfm) to recall, then tap the **Recall** button. The instrument displays the Reference waveform and opens the configuration menu.
5. Double-tap a math, reference, or bus badge to check or change that waveform's settings.
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

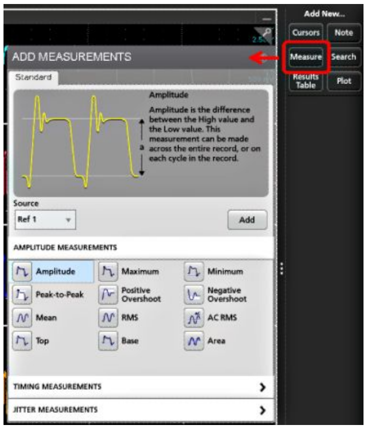
Use this procedure to select and add measurements.

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

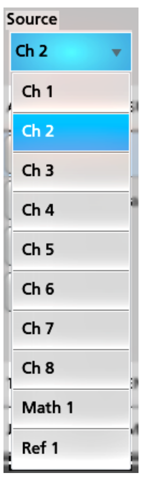


NOTE. *Waveforms do not need to be displayed to be used for measurements, as long as the channel or waveform badge is on the Settings bar and is acquiring the signal to measure.*

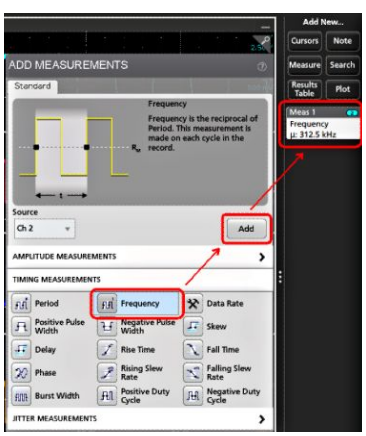
2. Tap the **Add New...Measure** button to open the **Add Measurements** configuration menu.



3. Tap the **Source** field and select the measurement source. All available sources that are valid for the measurement are listed.



4. Select from the configuration menu panels, such as Amplitude, Timing, and Jitter, to display measurements for those categories.
5. Select a measurement and tap **Add** (or double-tap the measurement) to add the measurement to the Results bar. The measurement badge is added immediately.



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.

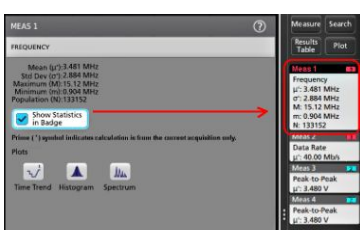
8. Tap outside the **Add Measurements** menu to close the menu.
9. To further adjust a measurement's settings, double-tap a measurement badge to open a configuration menu for that measurement. See *Configure a measurement* on page 80.
10. Tap the Help icon on the menu title for more information on settings.

Configure a measurement

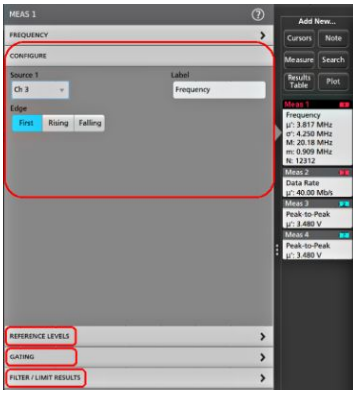
Use this procedure to add statistical readouts to the measurement badge, display plots for the measurement, and refine measurement parameters (configuration, global versus local scope of settings, gating, filtering, and so on).

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.

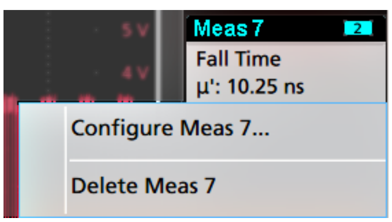


4. Use the available fields to refine the measurement conditions. Displayed fields depend on the measurement. Selection changes take effect immediately. Selection changes can also change fields in other panels.
5. Tap the Help button on the menu title for more information on this menu's settings.
6.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

Delete a Measurement or Search badge

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

1. Touch and hold the Measurement or Search 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.



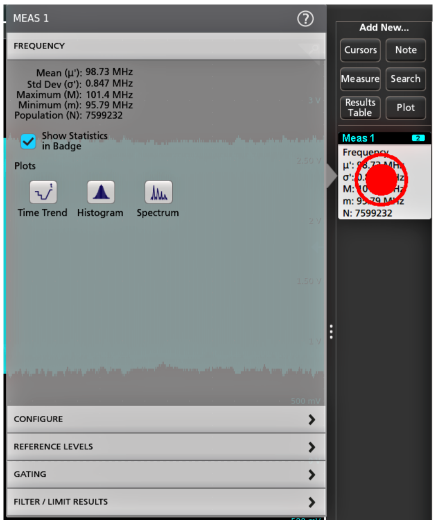
3. You can also delete a Measurement or Search badge by dragging it to the **Trash Can** icon, located at the bottom of the **Results** bar. When the Trash can icon and the badge turns red, lift your finger to delete the badge. You can use the mouse to drag and delete a badge.

***NOTE.** You can undo a measurement delete.*

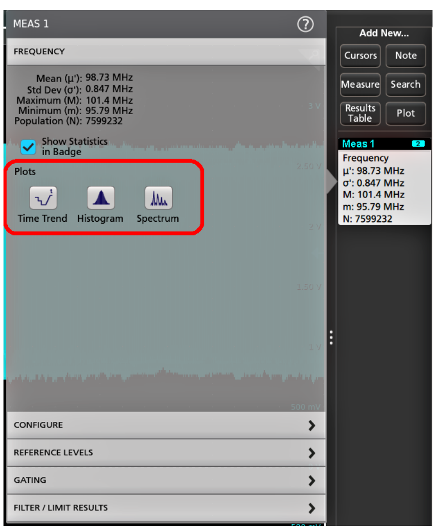
Add a plot of a measurement

Measurement plots let you graph the distribution of waveform datapoint occurrences (histogram), plot the frequency components (spectrum) of a waveform, show the time trend of a measurement, display an eye diagram, and other supported plots depending on the measurement. Available plots depend on the measurement.

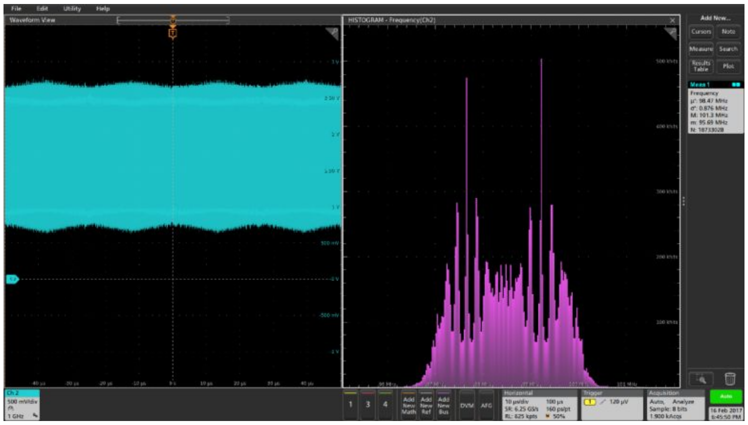
1. Double-tap a Measurement badge to open the **Meas** configuration menu.



2. Tap a **Plots** button to add that plot for the measurement to the screen.

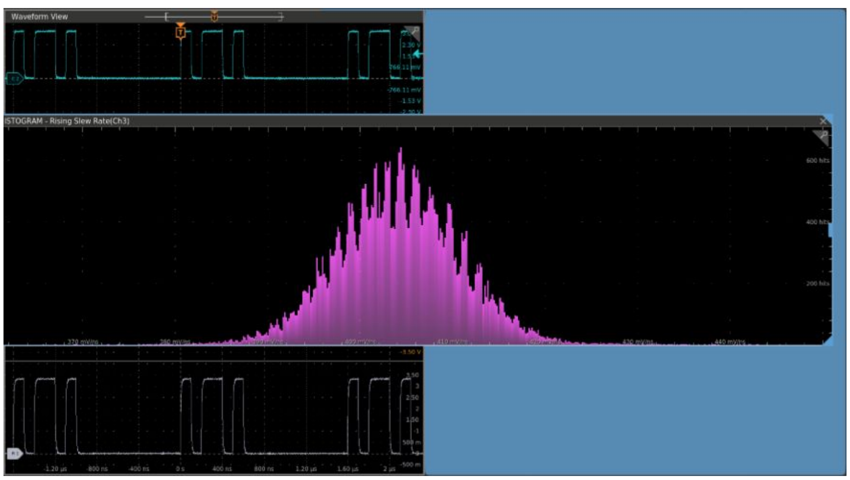


The following shows adding a Histogram plot.



You can add more than one plot to measurements (to different measurements or the same measurement). For example, you can add two histogram plots for the same measurement, set one to display the X-Axis with a Logarithmic scale, and the other plot to display the X-Axis with a Linear scale.

3. You can move plot windows by dragging the Plot view title bar to a new position. The blue background area moves to show where the plot will be located when you remove your finger from the title bar. You can also resize plot windows by selecting and dragging the Plot view border. You should use a mouse to do these operations, as it is easier to select and drag plots with a mouse.



4. Double-tap within a Plot view to open a configuration menu to set display characteristics. Tap the Help icon on the configuration menu title for more information on that menu's settings. Tap outside the menu to close the menu.

Display a Histogram plot

Use this procedure to display a histogram plot.

To display a histogram plot you must be taking a measurement.

1. Double-tap a measurement badge.

   The Measurement configuration menu is displayed.
2. Tap the **Histogram** plot button.

   The Histogram plot is displayed in a separate Plot view.
3. Double-tap in the plot view to open a configuration menu for that plot.

Display a Time Trend plot

Use this procedure to display a time trend plot.

To display a time trend plot you must be taking a measurement.

1. Double-tap a measurement badge.

   The Measurement configuration menu is displayed.
2. Tap the **Time Trend** plot button.

   The Time Trend plot is displayed. Time Trend plots are shown in the Waveform View rather than in a separate Plot view.

Display a Spectrum plot

Use this procedure to display a spectrum plot.

To display a time spectrum plot you must be taking a measurement.

1. Double-tap a measurement badge.

   The Measurement configuration menu is displayed.
2. Tap the **Spectrum** plot button.

   The Spectrum plot is displayed in a separate Plot view.
3. Double-tap in the plot view to open a configuration menu for that plot.

Display an XY or XYZ plot

Use this procedure to display an XY or XYZ plot.

1. Tap **Add New... Plot.**

   The Plot configuration menu is displayed.
2. If creating an XY plot, tap **XY**.
3. If creating an XYZ plot, tap **XYZ**.
4. Tap the **X Source** drop down and select the X Source from the list.
5. Tap the **Y Source** drop down and select the Y Source from the list.
6. If creating an XYZ plot, tap the **Z Source** drop down and select the Z Source from the list.
7. Tap **Add.**

   The plot is displayed in a separate Plot view.
8. 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 **Add New Math** 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. See *Math FFT plot configuration menu (Math waveform)* on page 270.

Display a Safe Operating Area (SOA) measurement and plot (option 5-PWR, SUP5-PWR, or 5-PS2 only)

Use this procedure to display a SOA plot. This is an XY plot with a mask.

To display an SOA power measurement and plot:

1. Tap the **Add New... Measurement** button.
2. Tap the **Power** tab.
3. Set the voltage and current sources for the measurement.
4. Tap the **Switching Analysis** panel.
5. Select the **SOA** measurement and tap **Add**.

The SOA measurement adds a Power measurement badge to the Results bar, and automatically adds the SOA plot to the screen.

6. Double-tap in the Plot view to open a configuration menu for that plot.

Display a power Switching Loss (SWL) Trajectory plot (option 5-PWR, SUP5-PWR, or 5-PS2 only)

Use this procedure to display a trajectory plot for a power measurement.

To display a trajectory power plot, you must be taking a Switching Loss measurement (Add New... Measurement > Power tab > Switching Analysis panel > Switching Loss measurement).

1. Double-tap the **Switching Loss** measurement badge to open the configuration menu.
2. Tap **Trajectory Plot** to add a new Trajectory Plot view to the screen.
3. Double-tap in the Plot view to open a configuration menu for that plot.

Display instantaneous power and energy plots from the Power Quality measurement. (option 5-PWR, SUP5-PWR, or 5-PS2 only)

Use this procedure to display a power/energy plot from the Power Quality measurement.

To display the power and energy plots you must be taking the Power Quality measurement.

1. Tap the **Add New... Measurement** button.
2. Tap the **Power** tab.
3. Set the voltage and current sources for the measurement.
4. Tap the **Input Analysis** panel.
5. Select the **Power Quality** measurement and tap **Add**.

   The measurement is added to the Results bar, and automatically adds **PQ: Power** and **PQ: Energy** math waveforms (plots) to the Waveform view. Math plots are shown in the Waveform View rather than in a separate Plot view.

   - PQ: Power displays the power waveform computed from the Power Quality input sources.
   - PQ: Energy displays the energy waveform computed from the Power Quality input sources. See *Waveform View configuration menu* on page 332.

Display a Harmonics Bar Graph plot for the Power Harmonics measurement (Option 5-PWR, SUP5-PWR, or 5-PS2 only)

Use this procedure to display a bar graph.

To display a harmonics bar graph, you must be taking a Harmonics measurement (Add New... Measurement > Power tab > Input Analysis panel > Harmonics measurement)

1. Double-tap a **Harmonics** measurement badge.

   The Measurement configuration menu is displayed.
2. Tap the **Bar graph** button.

The Bar graph is displayed in a separate Plot view.

3. Double-tap in the Plot view to open the configuration menu for that plot.

Add a Search

Use this procedure to set search criteria and mark a waveform where those events occur.

You can search on analog and digital signals, math waveforms, and reference waveforms. You can add searches to different waveforms and multiple searches to the same waveform. Prerequisite: Display the channel or waveform signal on which to search. The waveform must be displayed to create a search for it.

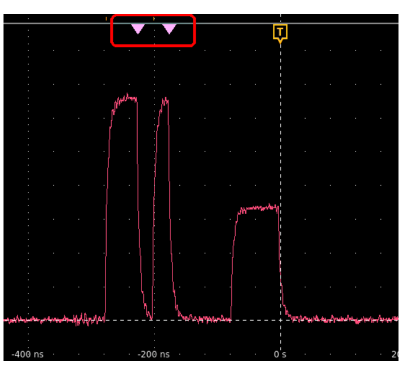
1. Display the channel or waveform signal on which to search. The waveform must be displayed to create a search for it.
2. Tap the **Add New...Search** button to open the Search configuration menu.

3. Use the configuration menu fields to set the search criteria in the same way that you would set for a trigger condition (select the **Search Type**, **Source**, and conditions on which to search).

NOTE. *You cannot search for sequential events (there is no Sequence search type).*

---

4. The searched waveform is marked with one or more triangles as soon as the search criteria becomes true. Each search uses a different color for its markers. The example image shows search criteria set to find positive pulse widths that are less than 70 ns wide.



5. To stop showing marks on a waveform, double-tap the **Search** badge and tap **Display** to **Off.**

6. To move the waveform to center marks on the display, push the **Run/Stop** front panel button to stop acquisition, single-tap a **Search** badge, and tap the < or > Navigation button.

NOTE. *Navigation buttons are only functional when the oscilloscope acquisition mode is set to* ***Stop****.*

This opens the **Zoom** mode and moves the waveform to the previous or next event mark on the waveform.

7. If available for a search, tap the **Min** or **Max** button to center the waveform in the display at the minimum or maximum value of the search events in the waveform record.

8. To return the instrument to normal acquisition mode, tap the **Zoom** icon in the upper right corner of the Waveform View to turn off **Zoom** mode, and push the **Run/Stop** front-panel button to set it to Run mode.

Change waveform view settings

Use this procedure to change the waveform display mode (Stacked or Overlay), 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. Tap the buttons in the **Display Mode** to toggle between **Overlay** and **Stacked** modes.

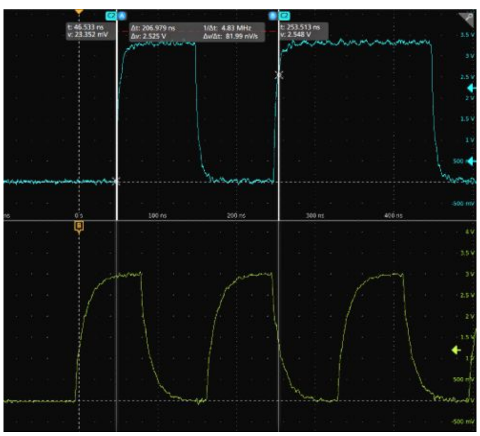
3. Use the other controls to set the waveform interpolation algorithm, waveform point persistence, style, and intensity, and graticule style and intensity.
4. Tap the **Help** icon on the menu title to open the Waveform View menu help topic for more information on the waveform view parameters.
5. Tap outside the menu to close the menu.

Display and configure cursors

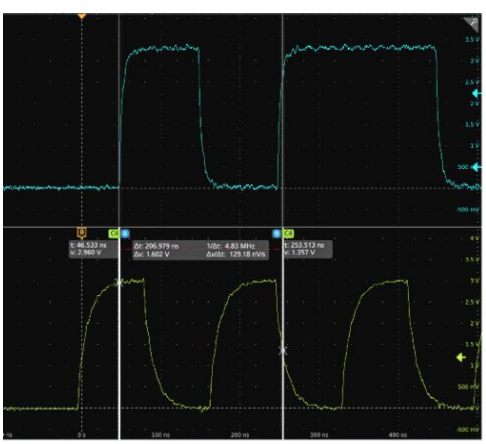
Cursors are on-screen lines that you can move to take measurements on specific parts of a waveform or plot, or between two different waveforms. Cursor readouts show both current position values and the difference (delta) between cursors.

1. Tap the waveform slice (in Stacked mode), or the channel or waveform badge (in Overlay mode) to which you want to add cursors.
2. Tap the **Add New...Cursors** button, or push the front-panel **Cursors** button.

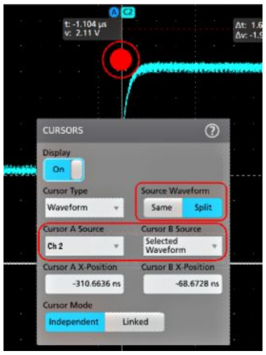
   The cursors are added to the display.



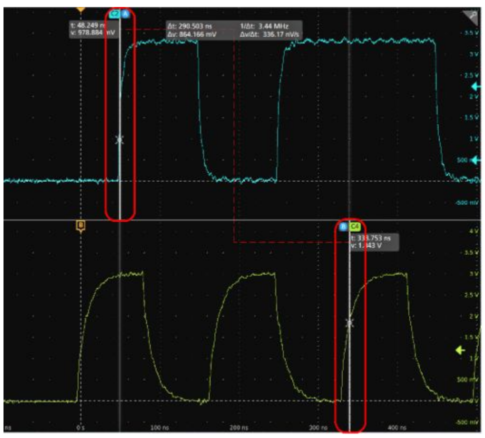
3. Use Multipurpose Knobs **A** and **B** to move the cursors, or touch and drag a cursor. Cursors show readouts that show position and difference measurements between the cursors.
4. To move the cursors to a different channel or waveform, just tap in that waveform graticule.



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.



6. To split the cursors between two waveforms, tap the **Source** field and select **Split** and select the source for each cursor.



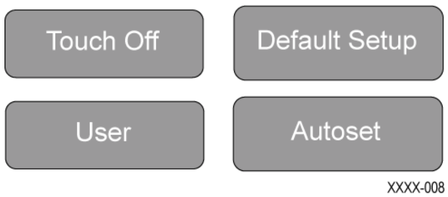
The cursors are moved to the specified waveforms.

7. Tap the Help icon on the menu title for more information on the menu settings.
8. To stop showing cursors, push the front panel **Cursor** button, press and hold to open the right-click menu and turn cursors off, or open the Cursors configuration menu and set Display to **Off**.

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).

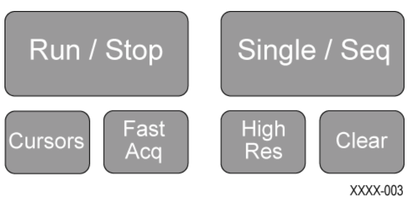


2. You can also select **File** > **Default Setup** to restore default settings.

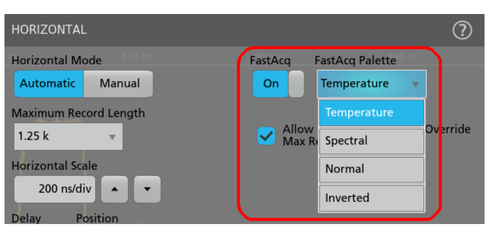
Using Fast Acq

Fast Acq (fast acquisition mode) reduces the dead time between waveform acquisitions, enabling the capture and display of transient events such as glitches or runt pulses. Fast acquisition mode can also display waveform phenomena at an intensity that reflects their rate-of-occurrence.

1. To use Fast Acquisition mode, double-tap the **Horizontal** badge and tap **Fast Acq** to toggle Fast Acq to On. You can also push the **Fast Acq** front-panel button.



2. To display waveform phenomena at an intensity that reflects their rate-of-occurrence, double-tap the Horizontal badge, tap the **Fast Acq Palette** field, and select a display palette from the drop-down.



3. View the waveform to find glitches, transients, or other random events. When you have identified an anomaly, use the advanced trigger system to capture the event of interest for further analysis.

If Fast Acquisitions mode is on and you attempt to activate a feature that conflicts with this mode, Fast Acquisitions mode will be inhibited. When the conflicting feature is turned off, in most cases, Fast Acquisitions will resume.

Add a note to a view

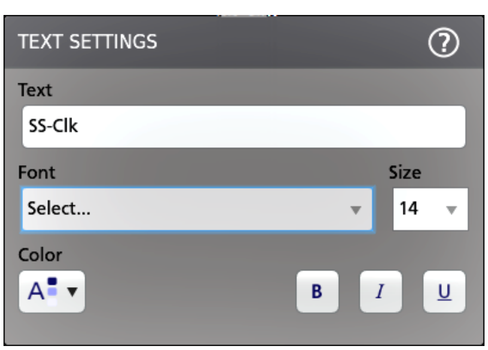
Tap the **Add New... Note** button to add text labels to your waveform and plot views. Use a mouse and keyboard to more easily edit and position a note.

1. Tap the **Add New... Note** button. The oscilloscope adds a text placeholder object called Edit text to the Waveform view.

NOTE. *The default note field wraps text at about 15 characters, and does not autowrap note text once the text is entered. If you need longer notes, resize the note field before entering text. Use a mouse to select the note and select a corner to resize the note field to a larger size.*

---

2. Double-tap on the note text top open the Text Settings configuration menu.



3. Tap in the **Text** field and use a keyboard to enter the note text, or double-tap in the Text field and use the on-screen keyboard to enter the note text.

NOTE. *You can enter only one row of text in the Text field of the Text Settings configuration menu (using an attached keyboard). You can enter multiple rows using the on-screen keyboard entry field.*

---

4. If you used the on-screen keyboard to enter text, tap the **Enter** button on the keyboard to close the keyboard and show the note on the screen.
5. Use the other Text Settings configuration menu controls to set font type, size, color and other characteristics.
6. Tap outside the Text Settings menu to close it.
7. To move a note, touch and drag the note text to a new position.

Delete a Note

Tap and hold on a note text to open a menu to delete that note.

1. Touch and hold on the note text you want to delete (or right-click with the mouse). The oscilloscope opens the right-click menu.
2. Select **Delete.** The note is deleted immediately.

NOTE. *You can select **Edit > Undo** on the menu bar to restore the note.*

Acquiring digital signals

Connect a TLP058 FlexChannel digital logic probe to any oscilloscope input channel. Connect the logic probe inputs to the DUT (see the probe instructions). Then use the following topics to set up, acquire and display digital signals.

- *Connect and set up digital signals* on page 95
- *Add a serial bus to the Waveform view* on page 96
- *Add a parallel bus to the Waveform view* on page 98

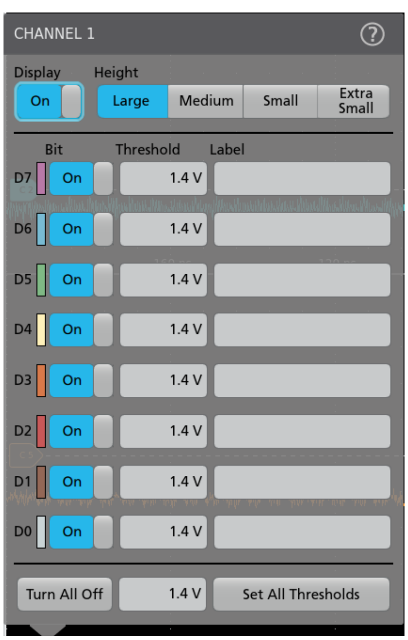
Connect and set up digital signals

Use the digital Channel configuration menu to set up the digital channels to acquire signals.

Digital channel configuration menus are available only if a FlexChannel-supported digital logic probe is attached to the oscilloscope.

CAUTION. *To prevent damage to the instrument always wear an antistatic wrist strap when making connections to the instrument and DUT. Always observe the maximum input voltage ratings for input connectors.*

1. Connect the FlexChannel logic probe to the instrument. The digital signal waveforms are opened on the screen.
2. Connect the probe to the signal sources. Use the accessories in the Tektronix Probe accessory Kit (shipped with the probe) to connect to your DUT.
3. Push the Menu button on the logic probe to open the Channel digital configuration menu. Set up digital channels to match your digital logic requirements.



4. Tap **Display** to toggle the digital channel group On or Off. Doing this closes the menu and removes the Digital channel badge from the Settings bar.
5. To change the displayed height of the digital channels, tap a **Height** button. The Height settings are only available when the display mode is set to Overlay in the Waveform View configuration menu.
6. Tap a **Bit** control to toggle individual digital channel bits On or Off and remove them from the displayed logic waveform.
7. Tap a **Threshold** control and use multipurpose knob A to set individual channel threshold levels. You can also set the threshold by double-tapping a control and setting the threshold using the virtual keypad.
8. Use the **Label** field to enter a custom label for digital channels. Tap the field and use a keyboard to enter label text. Or double-tap on the field and use the virtual keyboard to enter label text.
9. Tap **Turn All Off** to turn all digital bits Off. Doing this closes the menu and removes the Digital channel badge from the Settings bar.
10. To set all thresholds at once, enter a threshold value in the adjacent field and tap **Set All Thresholds**.

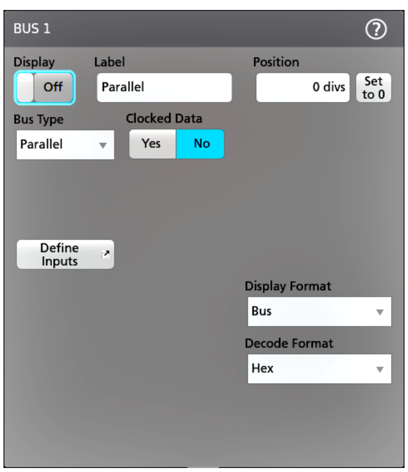
Add a serial bus to the Waveform view

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

This instrument supports the following buses: Parallel, SPI, I2C, USB, RS232, CAN, LIN, FlexRay, Ethernet, and Audio. All serial bus functions are options that must be purchased and installed before they are available in the menu.

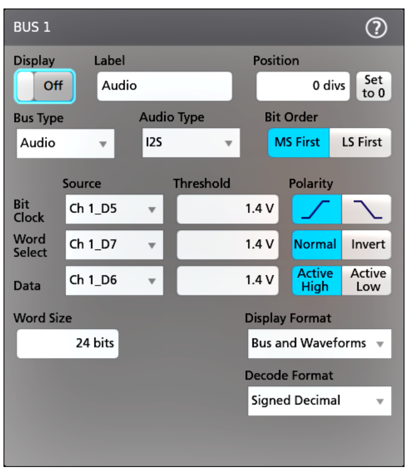
Use a Bus configuration menu to define a bus from which to acquire, decode, and display data. You can set up serial buses for the SPI, I2C, USB, RS232, CAN, LIN, FlexRay, Ethernet, and Audio serial data standards. You can also set up clocked or unclocked parallel buses.

1. Tap the **Add New Bus** button on the Settings bar to add a Badge bus to the Settings bar, add a bus waveform to the screen, and open the Bus configuration menu. The default bus type is parallel.

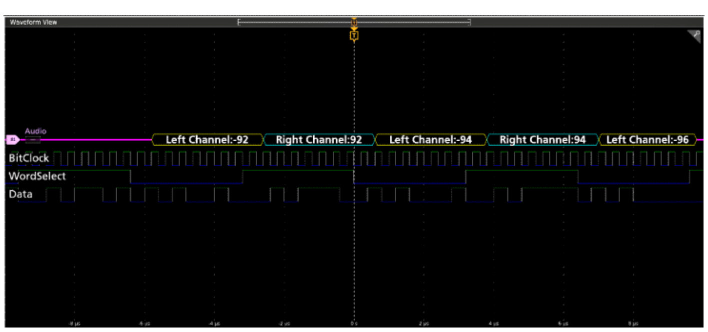


2. Tap **Bus Type** and select the bus type from the drop-down list.

3. Use the fields and controls to select the signal sources, thresholds, other parameters, and the output format. The following example shows the settings for an Audio I^2C serial bus.

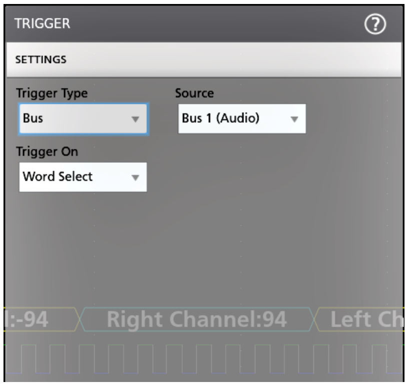


The decoded bus is updated on the screen as you make changes to the settings.



4. Tap outside of the Bus configuration menu to close it.

5. Double-tap the **Trigger** badge and use the Trigger configuration menu to trigger on a specific condition in the bus.



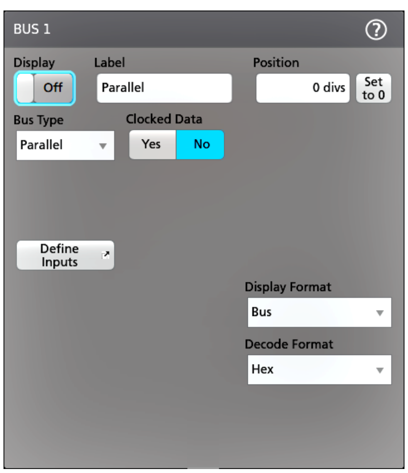
6. For more information on serial bus settings, tap the Help button on the Bus configuration menu.

Add a parallel bus to the Waveform view

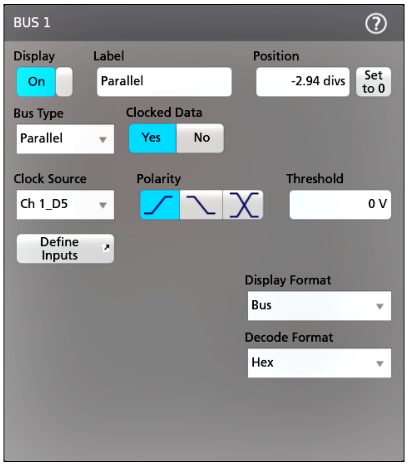
Use this procedure to add a parallel bus to the Waveform view.

When you acquire data from a Parallel bus, you can set up the bus to be clocked or unclocked. If the bus is not clocked, the instrument acquires all data from the parallel bus at the sample rate of the instrument.

1. Tap the **Add New Bus** button on the Settings bar. This adds a Bus badge bus to the Settings bar, adds a bus waveform to the screen, and opens the Bus configuration menu. The default bus type is parallel.



2. If setting up a clocked bus:

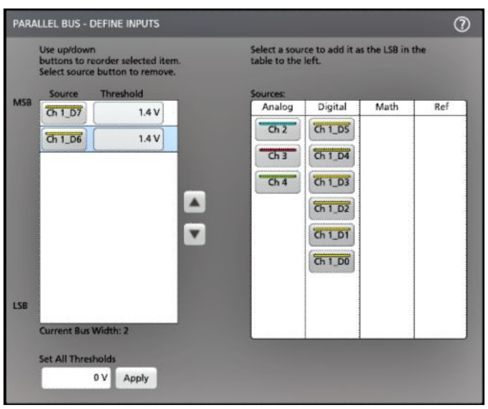


a. Set Clocked Data to **Yes**.

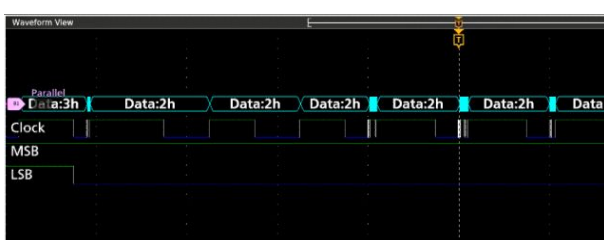
b. Tap the **Clock Source** field and select the source for the parallel bus clock signal.

c. Tap the **Polarity** and **Threshold** controls and set the clock signal transition to detect and threshold level, respectively.

3. Tap **Define Inputs** and select the signal sources for the parallel bus. Signal sources can be analog, digital, math, or reference. Tap a signal in the Sources list to add it to the bus list on the left.



The bus waveform updates as you make changes on the configuration menu. Tap the + symbol next to the waveform handle to turn on and off showing the signals associated with the bus waveform.



4. Use the rest of the fields and controls in the configuration menu to set up the parallel bus parameters(label, position, display and decode formats).

5. Tap outside of the Bus configuration menu to close it.
6. To get a stable triggered waveform, double-tap the **Trigger** badge, set the Trigger Type to **Bus**, select the bus Source to the parallel bus you just set up, and enter the data condition on which to trigger in the **Data** field.
7. For information on parallel bus menu settings, tap the Help button on the Bus configuration menu.

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. Use the following links for more information on advanced triggering.

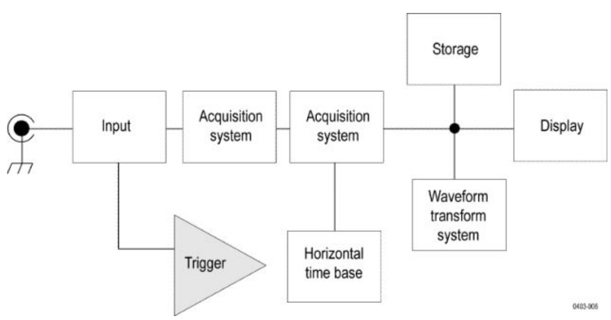
- *Triggering concepts* on page 101
- *Set Trigger Holdoff* on page 103
- *Trigger on sequential events (A and B triggers)* on page 103
- *Set up trigger on a parallel bus* on page 104
- *Set up trigger on a serial bus* on page 104

Triggering concepts

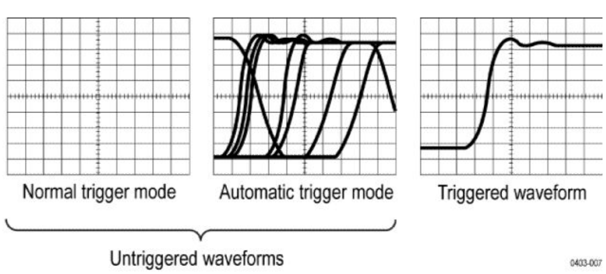
Overview

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.



Triggers help you 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).

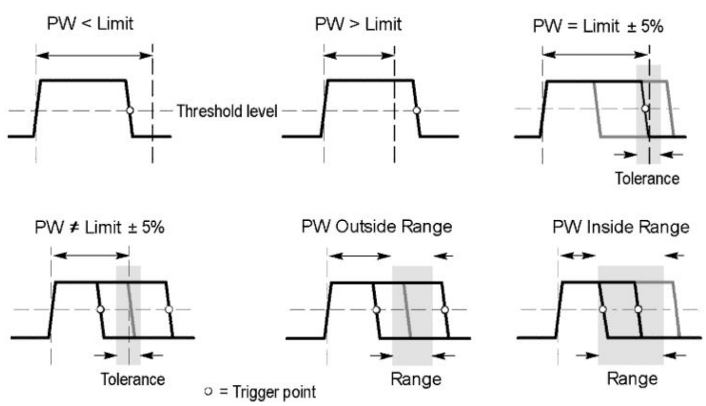
When a trigger event occurs, the instrument starts acquiring samples to build the posttrigger portion of the waveform record (displayed after, or to the right of, the trigger event). Once a trigger is recognized, the instrument will not accept another trigger until the acquisition is complete and the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

Trigger 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 set a pulse width trigger:

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

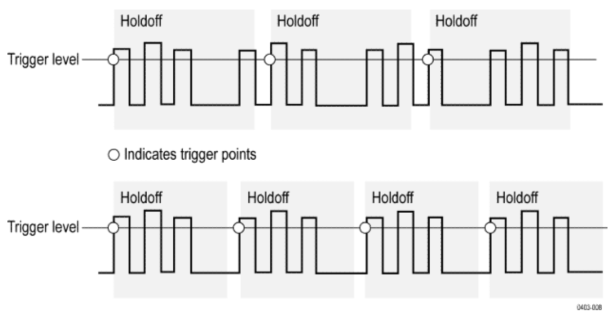


5. Set the pulse width time constraints:
   a. For all trigger when conditions except Outside Range or Inside Range, tap the **Time Limit** field and use the assigned multipurpose knob to set the pulse width time condition to meet.
   b. For Outside Range or Inside Range conditions, tap the **High Time 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.
8. To further refine the trigger conditions, set Logic Qualification to On and tap the **Define inputs** button to define the logic condition of any other signals that must be true at the same time as the pulse width trigger event is true.

Set Trigger Holdoff

Trigger Holdoff sets the time, after triggering on an event, that the instrument waits before detecting the same trigger event to start the next acquisition.

Setting the correct holdoff time is important to get a stable trigger. The longer holdoff time for the top waveform causes unstable triggering. The shorter holdoff set for the bottom waveform only triggers on the first pulse in the burst to remedy the unstable trigger.



Prerequisite: You have set up a trigger event for a signal.

1. Double-tap the **Trigger** badge on the Settings bar to open the Trigger configuration menu.
2. Tap the **Mode & Holdoff** panel.
3. To set a specific holdoff time, tap **Time**, then tap **Holdoff Time** and use the assigned multipurpose knob to specify a holdoff time. Or double-tap the field and use the virtual keypad to enter a holdoff time.
4. To delay the trigger a random amount of time between triggers, tap **Random**.

Trigger on sequential events (A and B triggers)

Use the A and B Trigger Events to trigger on a second event after a first event occurs.

Set up sequential triggering

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Sequence**.
3. Set up the A Trigger Event:
   a. Tap the **A Trigger Event** button to open the A Trigger Event menu.
   b. Tap **Trigger Type** and select a trigger type from the drop-down list. This example uses Edge.
   c. Tap **Source** and select the A event trigger source.
   d. If displayed, tap **Coupling** and select the trigger coupling.
   e. Tap **Level** and set the desired trigger level using the multipurpose knob. Or double-tap the field to set the value using the virtual keypad.
   f. Tap a **Slope** button to select the slope of the signal on which to trigger (Rise, Fall, Either).
   g. Tap anywhere outside the A Trigger Event menu to close it.
4. Set up the B Trigger Event:
   a. Tap the **B Trigger Event** button in the main Trigger configuration menu to open the B Trigger Event menu.

b. Tap **Trigger Type** and select a trigger type. This example uses Runt.

c. Tap **Source** and select a trigger source.

d. Tap **Trigger When** and select the condition on when to trigger on a runt signal.

e. Tap a **Polarity** icon to set the runt pulse polarity to detect (positive, negative, or either).

f. Tap the **Upper Threshold** and **Lower Threshold** fields and set the levels that a signal must cross to be considered a valid signal. Signals that don't cross both these thresholds are runt signals.

g. Tap anywhere outside the B Trigger Event menu to close it.

5. To trigger on a specific occurrence of the B trigger event:

   a. Tap After the A Trigger Event is found: **Trigger on the Nth Trigger Event** button in the main Trigger menu.

   b. Tap **Where N is:** and use the multipurpose knob to set the oscilloscope to trigger on the Nth occurrence of a B trigger event.

6. To trigger on the B event after a specific time delay:

   a. Tap After the A Trigger Event is found: **Trigger on the 1st B Event** button.

   b. Tap **After a Delay of:** and use the multipurpose knob to set the desired delay time to wait before detecting and triggering on a B trigger event. You can also double-tap the field and use the virtual keypad to enter a delay time.

Set up trigger on a parallel bus

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

Use this procedure if you have already created a parallel bus.

1. Double-tap the **Trigger** badge.
2. Tap the **Trigger Type** field and select **Bus** from the list.
3. Tap the **Source** field and select the parallel bus on which to trigger.
4. Tap either the **Binary** or **Hex** Data boxes to enter the parallel bus data value, in either Binary or Hexadecimal format, on which to trigger. The number of bits shown depends on the number of sources (channels) in the parallel bus.

   a. Use multipurpose knob A to select the digit or digits to change.

   b. Use multipurpose knob B to change the value of the selected digits.

Set up 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. Serial buses require purchasing and installing serial bus options. See *Serial bus and trigger options* on page 11.

1. Double-tap the **Trigger** badge on the Settings bar.
2. Tap **Trigger Type** and select **Bus** from the list.
3. Tap **Source** and select a serial bus from the list.
4. Tap **Trigger On** and select what to trigger on from the list. The displayed fields and controls depend on the bus type and Trigger On selection. Use these fields to trigger on a specific bus condition.

Trigger on an external signal (MSO58LP only)

Use this procedure to trigger an MSO58LP instrument from an external signal connected to the AUX input.

1. Double-tap the **Trigger** badge on the Settings bar.
2. Tap **Trigger Type** and select **Edge** from the list.
3. Tap **Source** and select **Aux**.

NOTE. *Aux is only available for the Edge trigger type.*

---

4. Set the values for **Coupling**, **Level**, and **Slope** to trigger on the Aux connector signal.

Setting waveform display parameters

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

Use the following topics for more information on setting display parameters.

Set waveform display mode (Stacked or Overlay)

Use this procedure to change the waveform display mode (Stacked or Overlay).

1. Double-tap on an open graticule area to open the Waveform View configuration menu.
2. Tap the **Display Mode** buttons to toggle between **Overlay** and **Stacked** modes.

   In stacked display mode, each waveform is stacked vertically in separate graticule slices. This is the default display mode.

   In overlay display mode, all waveforms are displayed in a single graticule (traditional waveform view).

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.

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.
   - **Off** disables display persistence.
   - **Auto** lets the oscilloscope automatically determine a persistence time for you.
   - **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.
   - **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.

   Time provides a vertical grid of time marks, 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.

   NOTE. *Changing the graticule intensity also changes the intensity of vertical scale readouts and horizontal time readouts on the screen. Graticule intensity does not change the intensity of note text on the screen.*

Zooming on waveforms

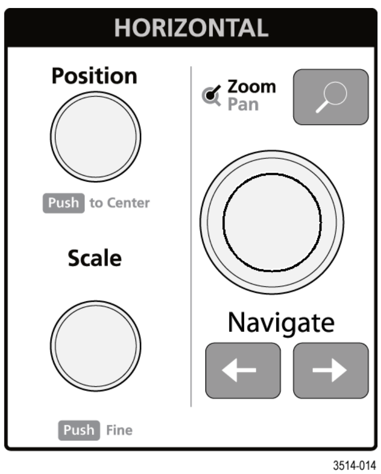
Use the zoom tools to magnify waveforms to view signal details.

Turn on Zoom mode

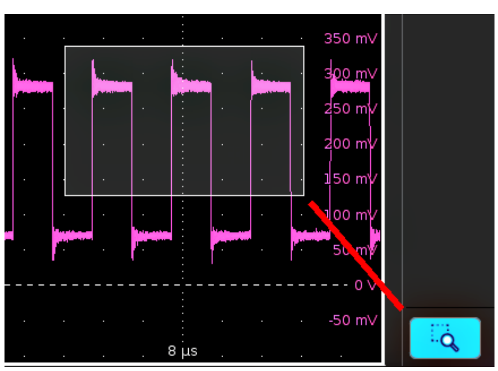
Zoom mode lets you look at a portion of your waveform in greater detail.

To enable Zoom mode, use one of the following methods:

1. Push the front-panel **Zoom** button.

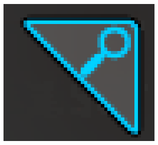


2. Tap the **Draw-a-Box** button.



This feature lets you quickly draw a box around an area of interest in the Waveform or Zoom Overview on which to zoom. Drawing a box immediately puts the oscilloscope view into zoom mode. Tap the Draw-a-Box button, then touch and drag in the Waveform view to draw a box on the area of interest. You can double-tap the Draw-a-Box button to keep it enabled, so that you can draw boxes in different areas of the Zoom Overview, or draw boxes to further zoom in on a waveform in the Zoom View.

3. Tap the Zoom icon in the corner of Waveform and Plot views.



4. For more information see *The Zoom user interface elements* on page 52.

Using Wave Inspector front-panel controls for zoom

Use the Wave Inspector controls to increase or decrease the area of the zoom box and to control the part of the waveform shown in the main Zoom view.

1. Push the front-panel **Zoom** button to open Zoom mode. Push the Zoom button again to exit zoom mode.
2. Turn the **Zoom** knob (center knob) to increase or decrease the horizontal area of the zoom box in the Zoom Waveform Overview, which in turn controls the amount of the waveforms shown in the main Zoom view.
3. Turn the **Pan** knob (outer knob) to move the Zoom box left or right in the Zoom Waveform Overview, which in turn controls the part of the waveform shown in the main Zoom view.
4. For information on the Zoom interface, see *The Zoom user interface elements* on page 52.

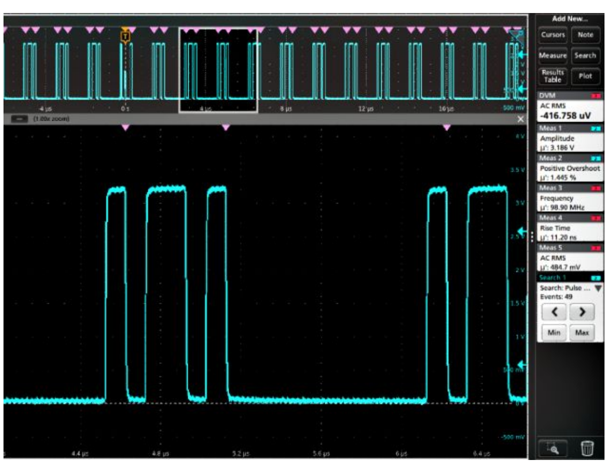
Zoom mode and Searches

Use Zoom and Searches to find events of interest

Searches provide a way to mark a waveform event or events for reference. You can set marks automatically with search criteria such as particular edges, pulse widths, runts, logic states, rise/fall times, setup and hold, and bus data types.

When in Zoom mode, you can use the front-panel left and right arrow buttons to position the waveform to previous or next search marks on the waveform.

You can also tap a Search badge and use its navigation buttons to position the waveform to previous or next search marks.



For more information on Searches, see *Add a Search* on page 87.

Customizing measurements

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

To customize measurements, double-tap a Measurement badge in the Results bar to open the *Measurement configuration menu overview* on page 177.

See the following topics for more information.

- **Label** your measurement to clarify documentation. See *Label a measurement* on page 113.
- **Reference levels** determine how time-related measurements are taken. See *Set measurement reference levels* on page 114.
- **Gating** confines the measurement to a certain portion of a waveform. See *Set measurement gates* on page 115.
- **Filters** control the band pass of a measurement. See *Set measurement filters* on page 115.
- **Limiting** the results you can specify a minimum or maximum value. See *Set measurement limits* on page 116.

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 overview* on page 177.

Prerequisite: To set measurement reference levels you must be taking a measurement. See *Add a measurement* on page 78.

1. Double-tap a **Measurement** badge in the Results bar to open the *Measurement configuration menu*.
2. Tap the **Configure** panel.
3. Use one of the following methods to enter label text:
   - Tap the **Label** field and enter the label text using a keyboard and press the keyboard Enter key to add the label text to the Measurement badge.
   - 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 overview* on page 177.

Prerequisite: To set measurement reference levels you must be taking a measurement. See *Add a measurement* on page 78.

1. Double-tap a Measurement badge.

   The Measurement configuration menu is displayed.
2. Tap the **Reference Levels** panel.
3. Select either Global (default) or Local:
   - **Global** causes changes in this panel to be updated in all other measurements that also have Global selected in this panel.

     When switching from Global to Local:
     - If a specific measurement has not been set to Local before then no changes are made to any of the values. You can update the parameters.
     - If a specific measurement has been set to Local before, then changed back to Global and then again switched to Local, the last used Local values are shown.
   - **Local** causes changes in this panel to only effect this measurement. When switching from Local to Global, all fields will update to the current Global parameters.
4. 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.
5. 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.
6. Touch **Hysteresis** and use the assigned multipurpose knob to set the value.
7. 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 overview* on page 177.

To set measurement gates you must be taking a measurement. See *Add a measurement* on page 78.

1. Double-tap a measurement badge to open the Measurement configuration menu.
2. Tap the **Gating** panel.
3. Tap **Gating** and select either Global or Local gating:
   - **Global** causes changes in this panel to be updated in all other measurements that also have Global selected in this panel.

     When switching from Global to Local:

     - If a specific measurement has not been set to Local before then no changes are made to any of the values. You can update the parameters.
     - If a specific measurement has been set to Local before, then changed back to Global and then again switched to Local, the last used Local values are shown.
   - **Local** causes changes in this panel to only effect this measurement. When switching from Local to Global, all fields will update to the current Global parameters.
4. 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.
   - **Logic** takes measurements only when the logical state of a specified waveform is true. If Logic gating is selected, continue with the following steps.

   a. Tap the **Source** field and select the source from the list.

   b. Tap the **Threshold** field and use the assigned multipurpose knob to set the desired threshold.

   c. Tap the **Hysteresis** field and use the assigned multipurpose knob to set the desired hysteresis.

   d. Tap **Active** and select either the High or Low active state. High sets the gating to take measurements when the specified waveform is an active High. Low sets the gating to take measurements when the specified waveform is an active Low
5. Tap anywhere outside the Measurement configuration menu to close it.

Set measurement filters

Use this procedure to set measurement filters.

Filters are set in the Filter/Limit Results panel of the Measurements configuration menu. See *Measurement configuration menu overview* on page 177.

To set measurement filters you must be taking a measurement. See *Add a measurement* on page 78.

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

3. Select either Global or Local.
   - **Global** causes changes in this panel to be updated in all other measurements that also have Global selected in this panel.

     When switching from Global to Local:
     - If a specific measurement has not been set to Local before then no changes are made to any of the values. You can update the parameters.
     - If a specific measurement has been set to Local before, then changed back to Global and then again switched to Local, the last used Local values are shown.
   - **Local** causes changes in this panel to only effect this measurement. When switching from Local to Global, all fields will update to the current Global parameters.
4. Tap the **High Pass Filter** field and select the type of filter from the drop down list.

   If you select a filter other than None, tap the **HPF Freq (F1)** field and set the frequency using the assigned multipurpose knob. Or double-tap the field and use the virtual keypad to enter a value.
5. Tap the **Low Pass Filter** field and select the type of filter from the drop down list.

   If you select a filter other than None, tap the **LPF Freq (F2)** field and set the frequency using the assigned multipurpose knob. Or double-tap the field and use the virtual keypad to enter a value.
6. Tap anywhere outside the Measurement configuration menu to close it.

Set measurement limits

Use this procedure to set measurement limits. Measurement limits let you set boundaries to eliminate nonrelevant values from a measurement.

Measurement limits are set in the Filter/Limit Results panel of the Measurements configuration menu. See *Measurement configuration menu overview* on page 177.

To set measurement limits you must be taking a measurement. See *Add Measurements configuration menu overview* on page 129.

1. Double-tap a Measurement badge in the Results bar to open the Measurements configuration menu.
2. Tap the **Filters/Limit Results** panel.
3. Tap **Limit Measurement Results** to let you restrict displayed measurements to only those that are within a specified range.
   - Tap the **Min Value** field and set the minimum acceptable measurement value using the assigned multipurpose knob. Or double-tap the field and use the virtual keypad to enter a value.
   - Tap **Max Value** and set the maximum acceptable measurement value using the assigned multipurpose knob. Or double-tap the field and use the virtual keypad to enter a value.
4. Tap **Limit Measurement Population** to let you set the number of measurement acquisitions (population) to acquire before stopping acquisitions for this measurement.
   - Tap the **Limit** field and set the population limit using the assigned multipurpose knob. Or double-tap the field and use the virtual keypad to enter a value.

NOTE. *If Limit Measurement Results and Limit Measurement Population are both On, the values in the Measurement badge are for the number of measurement acquisitions set in the population Limit field that were within the Limit Measurement Results settings.*

---

5. Tap anywhere outside the Measurement configuration menu to close it.

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. **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* ***User*** *button to immediately save the same type file again, without opening any menus.*

Save a waveform to a file

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

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

   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. You can save a single waveform or all waveforms
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 User 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 *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. Enable **Include Reference Waveforms** to include waveform files for all active waveforms in the setup file.
7. 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* ***User*** *button to immediately save the same type file again, without opening any menus.*

Save reports

Use this procedure to save reports.

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

   The Save As configuration menu opens.
2. Tap **Report** to open the Report 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 report file format from the list.
7. Tap **Append Report** to enable appending this report to a previous report that uses the same file name.
8. Double-tap **Comments** and use the virtual keyboard to add descriptive comments to the report.
9. Tap **Include Images and Annotations** to include screen images and annotations in the report.

10. Tap **Include Setup Configuration** to include the instrument settings data in the report.
11. Tap **Save** to save the report file 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 **User** button to immediately save the same type file again, without opening any menus.*

Save sessions

Use this procedure to save a session file. A session file contains instrument setup information and reference waveforms of active signals.

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

   The Save As configuration menu opens.
2. Tap **Session** to open the Session 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** to save the session data 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 **User** 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. There is no set limit to the number of reference waveforms that you can load and display.

1. Tap the **Add New Ref** button on the Settings bar.

   The *Recall configuration menu* opens showing the settings last used to recall a waveform file.
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. 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 (E -K).
   - 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.
4. 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.

   NOTE. *You can double-tap on a file name to immediately recall the file and close the menu.*
5. 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 (E -K).
   - 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.

NOTE. *You can double-tap on a file name to immediately recall the file and close the menu.*

---

6. Tap **Recall**

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

Recall a Session file

Use this procedure to recall (load) instrument settings and associated waveforms (as Reference waveforms) from a Session file.

1. Select **File > Recall** from the Menu bar to open the *Recall configuration* menu.
2. Tap **Session** to open the Session tab.

   The Recall configuration menu opens to show the settings last used to recall a session 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 (E -K).
   - 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.

NOTE. *You can double-tap on a file name to immediately recall the file and close the menu.*

---

6. Tap **Recall**.

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

Menus and dialog boxes

The Acquisition configuration menu

Use this configuration menu to set which data points are used to acquire waveforms, and enable automatically saving acquisitions to files.

To open the Acquisition menu, double-tap the **Acquisition** badge on the Settings bar.

The Acquisition menu fields and controls

Displayed fields and controls can change depending on menu selections.

Field or control	Description
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.
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. **Peak Detect** alternates between saving the highest sample in one acquisition interval and the lowest sample in the next acquisition interval. Useful for capturing fast and random events such as narrow waveform pulses. **High Res** applies unique finite impulse response (FIR) filters based on the current sample rate. This FIR filter maintains the maximum bandwidth possible for that sample rate while rejecting aliasing. The filter removes noise from the oscilloscope amplifiers and ADC above the usable bandwidth for the selected sample rate. Implementation of the filter in hardware, ahead of the trigger and storage, reduces trigger jitter and enables using Fast Acq mode while in High Res mode. High Res mode also guarantees at least 12 bits of vertical resolution. The number of bits of resolution is displayed in the Acquisition badge in the Settings bar. The High Res sample rate and record length settings are displayed in the Horizontal badge. High Res mode sets the maximum real time sample rate to ½ the maximum sample rate. **Envelope** acquires and displays a waveform record that shows the extremes in variations over several acquisitions. The instrument saves the highest and lowest values in two adjacent intervals (similar to the Peak Detect mode). Unlike Peak Detect mode, the peaks are gathered over many trigger events. **Average** acquires and displays a waveform record that is the average result of several acquisitions. This mode reduces random noise.
Single Sequence/Stop After	Enables stopping acquisitions after a specified number of acquisitions. Only works when using the Single/Sequence button.

Field or control	Description
Save on Trigger	Enables automatically saving waveform and/or screen image files when a trigger event occurs. Optimally used with single triggers or small sequences to reduce the number of files saved. Use the **Configure** button to set save location.
Configure	Opens the **Save On Trigger** menu to set the type of file to save and the save location. See *Save On Trigger menu* on page 124.
Timebase Reference Source	Sets the time base source used by the oscilloscope to acquire and measure signals. Internal sets the oscilloscope to use an internal reference signal. External sets the oscilloscope to use a 10 MHz signal connected to the rear-panel Ref In connector.
Fast Frame panel	Sets the Fast Frame acquisition mode. See *FastFrame Panel* on page 125

Save On Trigger menu

Use this configuration menu to enable automatically saving a waveform file, screen capture, or both when a trigger event occurs.

This menu is accessed from the Acquisition badge.

To open the Save on Trigger menu:

1. Double-tap the **Acquisition** badge.
2. Tap the Save on Trigger **Configure** button.
3. Set the location, type of files, and waveform source to save.

To save Files when a trigger event occurs, select the **Save on Trigger** box in the Acquisition configuration menu.

Save on Trigger menu fields and controls.

Field or control	Description
Save Location	The folder location to which to save the files. Use the Browse button to navigate to and select the folder. You can save files on the instrument or to an attached USB drive.
Browse	Opens the Browse Save On Location configuration menu. Use the configuration menu to navigate to and select a folder in which to save the files.
File Name	Use the default file name or enter a file name. Tap the arrow icon to list the file names of recently-saved files up to a maximum of 20.
Files to Save	Select the file type(s) to save (Waveform, Screen Capture, or both).
Format	Select the file format to which to save the waveform or screen capture.
Source	Select the signal source to save to waveform files. You can select All to save all displayed waveforms to individual files.

FastFrame Panel

Use this panel to enable FastFrame™ mode and select the number of frames to acquire.

The FastFrame panel is accessed from the Acquisition badge menu.

To open the Acquisition menu:

1. Double-tap the **Acquisition** badge.
2. Tap the **FastFrame** panel.
3. Use the fields to set the FastFrame parameters.

Save on Trigger menu fields and controls.

Field or control	Description
FastFrame	Enables or disables the FastFrame mode. When enabled (On), the screen shows the last acquisition made when FastFrame was enabled.
Number of Frames	Shows the number of frames acquired. Tap on the field and use the multipurpose knob set the number of frames to acquire, or double-tap on the field and use the virtual keypad to enter a frame count. The maximum number of frames you can capture is displayed in the frame information readout area.
frame information readout area	Displays frame acquisition information. The information changes depending on instrument settings, primarily the timebase, sampling rate and record length.
Summary Frame	Enables or disables creating and displaying a summary frame at the end of the frame record. When the current acquisition mode is either Sample or High Res, the Summary Frame shows an average of all frames. When the current acquisition mode is Peak Detect, the Summary Frame shows an envelope of all frames.

FastFrame usage guidelines.

- Define the trigger conditions to capture only the waveform, or waveform segment, in which you are interested, before enabling FastFrame mode.
- You can use FastFrame mode to acquire and compare frames for multiple signal inputs (analog, digital, math).
- You can use Zoom mode to view details of frames.
- Use **Single/Seq** triggering to acquire a single set of frames.
- While in FastFrame mode, measurements are taken across all frames, not just the currently selected frame. Measurement badges show the **‘** symbol to indicate that the values are from the current acquisition (all frames). Lack of a **‘** means the values are across many FastFrame acquisitions.
- While in FastFrame mode, statistics are accumulated across all frames within each acquisition as well as across acquisitions.
- To obtain very rapid statistical results on massive populations, use **Run/Stop** to keep acquiring frames and keep statistics on all of them. When the acquisition is stopped, the last frames acquisition is still in memory and the instrument has accrued statistical data for a large number of frames.
- Search works across all frames, not just the currently selected frame.
- In search mode, the Navigation buttons (Prev/Next arrow buttons) move from frame to frame as needed based on where the event is found.
- You can save FastFrame waveforms as reference waveforms using the normal Save As process. Saved FastFrame waveforms can be recalled as Reference waveforms.
- You can save and recall sessions with Fast Frame active. All frames are restored when the Session is recalled.
- If you display a regular (non-FastFrame) reference waveform, the Reference waveforms is always displayed regardless of the currently selected frame.

FastFrame badge

Use the FastFrame badge in the Results bar to select which frame to view and enable frame waveforms overlay mode. FastFrame works on all displayed analog, digital, and math waveforms.

The FastFrame badge is opened when you enable the FastFrame mode in the *FastFrame panel* of the Acquisition configuration menu.

FastFrame badge fields and controls.

Field or control	Description
Frame count	Shows the total number of frames captured at the top of the badge.
Selected Frame	Shows the selected frame in all displayed waveforms (analog, digital, and math). Use Multipurpose A knob to scroll through and select specific frames. To enter a specific frame number to display, use the Selected Frame field in the *FastFrame configuration menu*. Frame scrolling uses a wrap-around method. For example, when you reach the end of the frame list, scrolling further displays frame 1. If you have enabled overlay mode, the current frame's waveform is highlighted in blue in all displayed waveforms (analog, digital, and math).
Reference Frame	Selects a frame in the current frame acquisition that you want to use as a time reference against which to compare the selected frame. Displays a Delta readout that shows the difference between the Selected Frame and the Reference Frame. To directly enter a specific frame number to display, use the **Reference Frame** field in the *FastFrame configuration menu*. Only available when **Include Reference Frame in Badge** is selected in the FastFrame configuration menu.
Time stamps	Shows the time difference between frame one and the Selected frame. If **Include Reference Frame in Badge** is enabled, each frame readout area (Selected and Reference) in the badge shows the time difference between frame one and the Selected and Reference frames. When both Selected and Reference frames are enabled, a **Delta** time readout shows the time difference between the Selected frame and the Reference frame. If the difference time displayed goes beyond 10 seconds, then digits are dropped from the right as needed and the displayed timestamp is rounded. Note that this is just for UI display purposes. The full, precise value is stored and available.
Navigation buttons	Tap to display the previous or next frame. Navigation buttons are only displayed when acquisition is set to Stopped.
Summary	Displays the summary frame. the Summary frame button is only displayed when the Summary Frame button is enabled in the Acquisition menu FastFrame panel.

FastFrame badge configuration menu

Use the FastFrame configuration menu to set FastFrame overlay mode, enable reference frame timestamp readouts, and display FastFrame-related plots.

The FastFrame configuration menu is accessed from the FastFrame badge. To open the FastFrame configuration menu, double-tap the upper (readouts) area of the **FastFrame** badge.

FastFrame configuration menu fields and controls.

Field or control	Description
Frame count	Shows the total number of frames captured at the top of the badge.
Overlay Frames	Overlays all acquired frames for each displayed source (analog, digital, and math). If you have enabled overlay mode, the current frame's waveform is highlighted in blue in all displayed waveforms (analog, digital, and math).
Selected Frame	Sets or shows the current Selected frame number, and also shows the associated frame waveform in the Waveform View. Tap the field and use the Multipurpose knob A to scroll through the frame list. Frame scrolling uses a wrap-around method. For example, when you reach the end of the frame list, scrolling further displays frame 1. To enter a specific frame number, double-tap on the **Selected Frame** field and use the virtual keypad to enter a frame number.
Include Reference Frame in Badge	Enables or disables providing a Reference frame readout in the FastFrame badge. A Reference frame readout shows a difference timestamp between Frame one and the Reference frame, and also the delta between the Selected frame and the Reference frame timestamps. A Reference frame is not shown or highlighted in the Waveform View
Reference Frame	Set a Reference frame number. Tap the field and use the Multipurpose knob B to set a frame number. To enter a specific frame number, double-tap on the **Selected Frame** field and use the virtual keypad to enter a frame number. Only available when **Include Reference Frame in Badge** is selected.
Plot FastFrame Results	Opens and displays the selected plot type. **Timestamp Time Trend** plots the delta values from frame to frame for the current acquisition (full set of frames). The first data point is the delta time between frame 1 and frame 2. The second data point is the delta time between frame 2 and frame 3 and so on. It does not add data points with subsequent acquisitions.

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 **Add New... Measure** button in the **Analysis** controls area.

The **Add Measurements** configuration menu always opens on the **Standard** measurement tab. The listed tabs and measurements depend on the installed measurement options and the selected signal source.

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 overview* on page 177.

Add Measurements menu fields and controls

Field or control	Description
Measurement tabs	The tabs along the top organize measurements by their type. The Standard tab is the default set of measurements that are built in to the instrument. Other tabs are shown when you install measurement options. See *The Jitter tab (Advanced Jitter and Eye Analysis option 5-DJA, SUP5-DJA only)* on page 135. See *The Power tab (option 5-PWR, SUP5-PWR, or 5-PS2 only)* on page 139.
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. If the measurement requires more than one source (for example, Skew, Phase, or many Power measurements), the menu shows two source fields from which to select.
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.

Standard tab measurement panels

Panel	Description
Amplitude Measurements panel	Tap the panel bar to list the available amplitude measurements. Touch and drag the list to scroll through all measurements. See *Amplitude Measurements panel* on page 130.
Timing Measurements panel	Tap the panel bar to list the available time measurements. Touch and drag the list to scroll through all measurements. See *Timing Measurements panel* on page 132.
Jitter Measurements panel	Tap the panel bar to list the available standard jitter measurements. See *Jitter Measurements panel* on page 134. ***NOTE.*** *If you have installed Advanced Jitter and Eye Analysis (Option DJA), the Jitter Measurements panel is removed from the Standard measurement tab and is replaced with a Jitter tab at the top of the Add Measurements menu. See The Jitter tab (Advanced Jitter and Eye Analysis option 5-DJA, SUP5-DJA only) on page 135.*

Other tab measurements

Add Measurement tab	Description
Jitter	Advanced Jitter and Eye Analysis measurements (Option DJA). Provides triggers and measurements for advanced jitter and eye analysis. See *The Jitter tab (Advanced Jitter and Eye Analysis option 5-DJA, SUP5-DJA only)* on page 135.
Power	Advanced Power Analysis (Option 5-PWR, SUP5-PWR, or 5-PS2). Provides measurements for input analysis, output analysis, amplitude analysis, timing analysis, and switching analysis measurements. See *The Power tab (option 5-PWR, SUP5-PWR, or 5-PS2 only)* on page 139.

The Standard measurements tab

The Standard tab lists the default set of amplitude, timing, and jitter measurements that are included with the instrument.

To open the Standard measurements tab, tap the **Add New...Measure** button. The Add Measurements configuration menu opens on the Standard tab by default.

To add a measurement to the Results bar:

1. Select the signal source.
2. Tap a measurement panel:

   Amplitude Measurements panel on page 130

   Timing Measurements panel on page 132

   Jitter Measurements panel on page 134
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.

NOTE. *If you have installed Advanced Jitter and Eye Analysis (Option DJA), all jitter measurements are in the* ***Jitter*** *tab. See The Jitter tab (Advanced Jitter and Eye Analysis option 5-DJA, SUP5-DJA only) on page 135.*

Amplitude Measurements panel

The Amplitude Measurements panel lists available amplitude-related measurements that you can take on analog channel signals, math waveforms (time-domain), and reference waveforms. Amplitude measurements are not available for digital signals.

To open the Amplitude Measurements panel:

1. Tap **Add New... 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. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Amplitude	The difference between the Top value and the Base value. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
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. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Base [1]	The most common data point value below the midpoint of the waveform, over the measurement region. Base is used as the 0% value whenever low reference values are calculated, such as in rise time or fall time measurements. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Maximum	The maximum data point value. You can take this measurement on each cycle in the waveform record or across the entire waveform record.
Mean	The arithmetic mean of all data points over the measurement region. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Minimum	The minimum data point value. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Negative Overshoot[1]	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. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Peak-To-Peak	The absolute difference between the Maximum and Minimum amplitudes in the measurement region. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Positive Overshoot[1]	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. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
RMS	The true Root Mean Square (The square root of the mean (average) value of the squared function of the waveform data points.) You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Top[1]	The most common data point value above the midpoint of the waveform over the measurement region. Base is used as the 100% reference value whenever high reference values are calculated, such as in rise time or fall time measurements. You can take this measurement on each cycle in the waveform record or on the entire waveform record.

See also. *Measurement configuration menu overview* on page 177

[1] Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated.

Timing Measurements panel

Use the Timing Measurements panel to add timing-related measurements to the Results bar. Timing measurements can be taken on time-domain analog, math, and reference waveforms. Timing measurements can also be taken on some digital channel signals.

To open the Timing Measurements panel:

1. Tap the **Add New... Measure** button.
2. Tap the **Timing 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.

Timing Measurements panel.

Measurement	Description
Burst Width [2]	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.
Data Rate[2]	Data Rate is the reciprocal of Unit Interval. The measurement is taken on each bit of the waveform record.
Delay	The time difference between the mid reference (default 50%) amplitude point of two different waveforms. You can specify the signal edges to measure in the measurement's Configuration menu. The measurement requires two sources.
Duration N-Periods[2]	The time required to complete N cycles. A cycle is the time between two adjacent (same direction) crossings of the mid reference level. The measurement is taken on each cycle in the 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.
Falling Slew Rate	The rate of change (in volts/second) as an edge transitions from a high reference level to a low reference level. The measurement is taken on each cycle of the record in the measurement region.
Frequency[2]	The frequency of the waveform. Frequency is the reciprocal of Period (Frequency = 1/Period).
High Time	The time the signal remains above the Top reference level. The measurement is taken on each cycle in the waveform record.
Hold Time[2]	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.
Low Time	The time the signal remains below the Base reference level. The measurement is taken on each cycle in the waveform record.
Negative Duty Cycle[2]	The ratio of the negative pulse width to the signal period, expressed as a percentage. The duty cycle is measured on the first cycle in the measurement region.
Negative Pulse Width[2]	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[2]	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.
Phase[2]	The time difference (phase shift) between the specified signal edges of waveform source 1 and waveform source 2. The measurement is expressed in degrees, where 360° comprise one waveform cycle. This measurement requires two sources. The measurement is taken on each cycle of the waveform record.
Positive Duty Cycle[2]	The ratio of the positive pulse width to the signal period, expressed as a percentage. The duty cycle is measured on the first cycle in the measurement region.

2 This measurement can also be taken on digital signals.

Measurement	Description
Positive Pulse Width[2]	The distance (time) between the mid reference (default 50%) amplitude points of a positive pulse. The measurement is made on the first pulse in the 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.
Rising Slew Rate	The rate of change (in volts/second) as an edge transitions from a low reference level to a high reference level. The measurement is taken on each cycle of the record in the measurement region.
Setup Time[2]	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.
Skew[2]	The time between the specified Mid reference level crossing on one source to the closest Mid reference level crossing on the second source signal. The measurement is made on each cycle in the waveform record.
Time Outside Level	The time the specified signal remains above the Top reference level and/or below the Base reference level. The measurement is made on each occurrence in the waveform record.
Unit Interval[2]	The time difference between two successive bits. The measurement is taken on each bit of the waveform record.

See also. *Measurement configuration menu overview* on page 177

Jitter Measurements panel

The Jitter Measurements panel lists the standard jitter-related measurements that you can add to the Results bar. These jitter measurements are part of the Standard measurements that are provided by default.

To open the Jitter Measurements panel:

1. Tap the **Add New...Measure** button.
2. Tap the **Jitter 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.

NOTE. *If you have installed Advanced Jitter and Eye Analysis (Option DJA), the jitter measurements are moved to the* ***Jitter*** *tab of the* ***Add Measurements*** *menu. See The Jitter tab (Advanced Jitter and Eye Analysis option 5-DJA, SUP5-DJA only) on page 135.*

Jitter Measurements panel measurements.

Measurement	Description
Phase Noise	The RMS magnitude of all integrated jitter falling within a specified offset range of the fundamental clock frequency. This measurement is taken on the entire waveform record.
TIE	The difference in time between an edge in the source waveform and the corresponding edge in a recovered reference clock signal. The measurement is made on each waveform edge in the measurement region.

See also. *Measurement configuration menu overview* on page 177

The Jitter tab (Advanced Jitter and Eye Analysis option 5-DJA, SUP5-DJA only)

The Jitter tab lists advanced jitter, eye, amplitude, and timing measurements that you can add to the Results bar. The Jitter tab is only shown if you have purchased and installed Advanced Jitter and Eye Analysis (option 5-DJA or SUP5-DJA).

To open the Jitter measurements tab:

1. Tap **Add New... Measure** button.
2. Tap the **Jitter** tab.

To add a measurement to the Results bar:

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

Adding eye diagram plots for a jitter measurement. To display an eye diagram plot for a jitter measurement, double-tap the jitter measurement badge for which you want to show an eye diagram. If available, tap the **Eye Diagram** plot button.

Jitter Measurements panel (option 5-DJA, SUP5-DJA).

Measurement	Description
DCD	Duty cycle distortion. The peak-to-peak amplitude for the component of the deterministic jitter correlated with the signal polarity. The measurement is taken on the entire record.
DDJ	Data dependent jitter. The peak-to-peak amplitude for the component of the deterministic jitter correlated with the date pattern in the waveform. The measurement is taken on the entire record.
DJ	Deterministic jitter. The peak-to-peak amplitude of all timing errors that exhibit deterministic behavior. The measurement is taken across the entire record.
DJ-δδ	Dual-Dirac deterministic jitter. The deterministic jitter on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal-magnitude Dirac functions. The measurement is taken on the entire record.
F/2	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (date rate) divided by two. The measurement is taken on the entire record.
F/4	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (date rate) divided by four. The measurement is taken across the entire record.
F/8	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (date rate) divided by eight. The measurement is taken on the entire record.
J2	The total jitter at a bit error rate of 2.5e-3 (TJ@2.5e-3). The measurement is taken on the entire record.
J9	The total jitter at a bit error rate of 2.5e-3 (TJ@2.5e-3). The measurement is taken on the entire record.
Jitter Summary	Adds multiple jitter measurements to the measurement badge, and displays Bathtub, TIE Spectrum trend plot, TIE Histogram, Eye Diagram plots on the screen.
NPJ	Nonperiodic jitter. The portion of the BUJ (Bound uncorrelated jitter) that is random. BJU excludes DDJ, DCD, and RJ. The measurement is taken on the entire record.
Phase Noise	The RMS magnitude of all integrated jitter falling within a specified offset range of the fundamental clock frequency. The measurement is taken on the entire record.
PJ	Periodic jitter. The peak-to-peak amplitude of the uncorrelated sinusoidal components of the deterministic jitter. The measurement is taken on the entire record.
RJ	Random jitter. The RMS magnitude of all random timing errors following a Gaussian distribution. The measurement is taken on the entire record.
RJ-δδ	Dual-Dirac random jitter. The random jitter on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal-magnitude Dirac functions. The measurement is taken on the entire record.
SRJ	Sub-rate jitter. The composite jitter due to periodic components at ½, 1/4, and 1/8 of the data rate. The measurement is taken on the entire record.

Measurement	Description
TIE	Time Interval Error. The difference in time between an edge in the source waveform and the corresponding edge in a recovered reference clock signal. The measurement is taken on each waveform edge.
TJ@BER	Total error at a specified bit error rate. The predicated peak-to-peak amplitude of jitter that will only be exceeded with a probability equal to the bit error rate. The measurement is taken on the entire record.

Eye Measurements panel (option 5-DJA, SUP5-DJA).

Measurement	Description
Eye Height	The minimum vertical eye opening at the center of the recovered unit interval. The measurement is taken on the entire waveform record.
Eye High	The amplitude of a high (one) bit measured at a specified location within the recovered unit interval. The measurement is taken on each high bit in the waveform record.
Eye Low	The amplitude of a low (zero) bit measured at a specified location within the recovered unit interval. The measurement is taken on each low bit in the waveform record.
Eye Width	The minimum horizontal eye opening at the center of the recovered unit interval. The measurement is taken on the entire waveform record.
EyeHeight@BER	The predicted vertical eye opening that will be violated with a probability equal to the bit error rate. The measurement is taken on the entire waveform record.
EyeWidth@BER	The predicted horizontal eye opening that will be violated with a probability equal to the bit error rate. The measurement is taken on the entire waveform record.
Q-Factor	The ratio of the vertical eye opening to RMS vertical noise measured at a specified location in the recovered unit interval. The measurement is taken on the entire waveform record.

Amplitude Measurements panel (option 5-DJA, SUP5-DJA).

Measurement	Description
AC Common Mode (Pk-Pk)	The peak-to-peak of the common mode voltage of the two specified sources. The measurement is taken on the entire record.
Bit Amplitude	The difference between the amplitudes of the 1 and 0 bits surrounding a transition. The amplitude is measured over a specified part of at the center of the recovered time interval. The measurement is taken on each transition bit of the entire record (Mean) or on the entire record (Mode).
Bit High	The amplitude of a 1 bit, measured over a specified percent of the center of the recovered time interval. The measurement is taken on each high bit of the entire record (Mean) or on the entire record (Mode).
Bit Low	The amplitude of a 0 bit, measured over a specified percent of the center of the recovered time interval. The measurement is taken on each low bit of the entire record (Mean) or on the entire record (Mode).
DC Common Mode	The arithmetic mean of the common mode voltage of two sources. The measurement is taken on the entire record.
Differential Crossover	The voltage level of a differential signal pair at the crossover point(s). The measurement is taken at each crossover point in the record.
T/nT Ratio	The ratio of a nontransition bit voltage (second and subsequent bit voltage after a transition) to its nearest preceding transition bit voltage (first bit voltage after the transition). Bit voltage is measured at the interpolated midpoint of the recovered unit interval. The measurement is taken on each nontransition bit in the record.

Timing Measurements panel (option 5-DJA, SUP5-DJA).

Measurement	Description
SSC Freq Dev	The spread spectrum clock frequency deviation. This measurement enables a time trend plot of the spread spectrum clock modulation profile. The measurement is taken on each cycle of the entire record.
SSC Modulation Rate	The modulation frequency of a spread spectrum clock. The measurement is taken on each cycle of the entire record.

See also. *Measurement configuration menu overview* on page 177

The Power tab (option 5-PWR, SUP5-PWR, or 5-PS2 only)

The Power tab lists the power-related measurements that you can add to the Results bar. Power measurements include input analysis, amplitude analysis, timing analysis, switching analysis, and output analysis. The Power tab is shown only if you have purchased and installed the Advanced Power Analysis option (option 5-PWR, SUP5-PWR, or 5-PS2).

To open the Power Measurements tab:

1. Tap the **Add New...Measure** button.
2. Tap the **Power** tab.

To add a measurement to the Results bar:

1. Select the signal source/s.
2. Tap a measurement panel:

   Input Analysis Measurement panel

   Amplitude Analysis Measurement panel

   Timing Analysis Measurement panel

   Switching Analysis Measurement panel

   Output Analysis Measurement panel
3. Select the measurement. If the 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 to the Results bar.

Input Analysis Measurements panel (Option 5-PWR, SUP5-PWR, or 5-PS2).

Measurement	Description
Power Quality	Measures the Frequency, RMS values of the voltage and current, Crest Factors of the voltage and current, Real Power, Reactive Power, Apparent Power, Power Factor, and Phase Angle of the AC signal.
Harmonics	Plots the signal amplitudes at the fundamental line frequency and its harmonics. Measures the RMS amplitude and Total Harmonic Distortion of the signal. Plots the Harmonics Bar Graph.

Amplitude Analysis Measurements panel (Option 5-PWR, SUP5-PWR, or 5-PS2).

Measurement	Description
Cycle Amplitude	The difference between the Top value and the Base value. Measurement can be made across the entire record or on each cycle in the record.
Cycle Top	The most common data point value above the midpoint of the waveform over the measurement region. This measurement is made on each cycle in the record.
Cycle Base	The most common data value below the midpoint of the waveform. This measurement is made on each cycle in the record.
Cycle Peak-to-Peak	The difference between the Maximum and Minimum values in the measurement region. This measurement is made on each cycle in the record.
Cycle Maximum	The maximum data point. This measurement is made on each cycle in the record.
Cycle Minimum	The minimum data point. This measurement is made on each cycle in the record.

Timing Analysis Measurements panel (Option 5-PWR, SUP5-PWR, or 5-PS2).

Measurement	Description
Period	The time between two adjacent crossings of the Mid reference level (one cycle) of the waveform. This measurement is taken on each cycle of the waveform record or measurement region.
Frequency	Frequency is the reciprocal of Period (Frequency = 1/Period). This measurement is made on each cycle in the record.
Positive Duty Cycle	The ratio of the positive-pulse width to the signal period. This measurement is taken on each cycle of the waveform record or measurement region.
Negative Duty Cycle	The ratio of the negative-pulse width to the signal period. This measurement is taken on each cycle of the waveform record or measurement region.
Positive Pulse Width	The time the signal remains above the Mid-reference level. This measurement is taken on each cycle of the waveform record or measurement region.
Negative Pulse Width	The time the signal remains below the Mid-reference level. This measurement is taken on each cycle of the waveform record or measurement region.

Switching Analysis Measurements panel (Option 5-PWR, SUP5-PWR, or 5-PS2).

Measurement	Description
Switching Loss	The mean instantaneous power and energy in the turn-on, turn-off, and conduction regions of a switching device. Provides the SWL trajectory plot.
dv/dt	The rate of change (slew rate) of the voltage, as it rises from the Base reference level (R_B) to the Top reference level (R_T), and as it falls from the Top reference level (R_T) to the Base reference level (R_B).
di/dt	The rate of change (slew rate) of the current, as it rises from the Base reference level (R_B) to the Top reference level (R_T), and as it falls from the Top reference level (R_T) to the Base reference level (R_B).
SOA	An X-Y plot of switching device voltage and current. The SOA mask testing provides pass/fail testing to component specifications.

Output Analysis Measurements panel (Option 5-PWR, SUP5-PWR, or 5-PS2).

Measurement	Description
Line Ripple	The RMS and peak-to-peak measurements of the line frequency portion of the AC signal.
Switching Ripple	The RMS and peak-to-peak measurements of the input signal.

See also. *Power measurement configuration menu overview*

Add Plot configuration menu

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

These plots differ from the Measurement plots (added from a Measurement configuration menu), which plot that measurement's value, not the signal source value(s).

To open the **Add Plot** configuration menu, tap the **Add New... Plot** button.

To add a plot:

1. Select a plot type.
2. Select the signal source or sources.
3. Tap **Add**. You can also double tap a plot to add it to the screen. The plot is added to the screen.

Add Plot menu fields and controls

Field or control	Description
Sources	Sets the input sources for the plot. The number of source fields listed depends on the plot.
XY, **XYZ**, **Eye Diagram**	Select the plot type to add to the screen. **Eye Diagram** adds a TIE measurement badge to the Results bar and adds the eye diagram to the screen.
Add	Adds the selected plot to the screen. You can also double tap on a plot type to add it to the screen.

Configuring plots

Each plot has a menu that lets you configure that plot's settings. See *Plot configuration menus* on page 267.

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, or bus decode values, to the screen in a spreadsheet-like format.

To open the **Add Results Table** configuration menu, tap the **Add New... Results Table** button.

To add a result table to the screen:

1. Select a table type.
2. Tap **Add**. You can also double tap a plot to add it 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.
Harmonics	Displays a table of the Harmonics measurement results (Option 5-PWR, SUP5-PWR, or 5-PS2 only).
Add	Displays the selected table type. You can add as many tables as you want on the screen.

Results Tables operations overview

Results Tables (Measurement or Bus Decode) list all active measurements and their statistics, or bus decode activity, in a spreadsheet format that lets you quickly compare values or save the results to a report.

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.
- If you add or delete a bus to the Settings bar, that bus is automatically added to or deleted from an existing Bus Decode Results table.
- Each bus in a Bus Decode Results table has its own tab. Tap a tab show the decode results for that bus.
- Selecting a Bus in the Bus Decode Table configuration menu does not select and display the tab for that bus.
- To scroll up and down in a results table, tap the table and use Multipurpose knob A to scroll and select table rows.
- To move a column, touch and drag the column title to a new position in the table.
- 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.
- To resize a column width, use the mouse to position the cursor on the column name border to change, then click and drag the column border to resize that column.
- To configure or delete a single measurement in a Measurement Results table, touch and hold on a table row to open a right-click menu that lets you configure or delete that measurement.
- You cannot sort the contents of a column.

See also. *Measurement Table configuration menu* on page 142

Bus Decode Results table configuration menu on page 144

Harmonics Results table configuration menu (Option 5-PWR, SUP5-PWR, or PS2 only) on page 145

Measurement 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 Table configuration menu, double-tap anywhere in a Measurement Results table.

Measurement 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 column.
Set to Defaults	Sets all columns to show 5 digit readout resolution.
Other panel	
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.
Cycle-to-Cycle Variation	For measurements where this applies, this function allows you to see the mean or average variation between measurements performed on each available cycle in the waveform.
Row Height Optimized For	Select Touch to increase table row height by ~75% for easier touch manipulation. Uses more space on the screen. Select Mouse to decrease row height. Uses less space on the screen. Mouse is the default setting.
Save Table	Opens a menu with which to save the results table data to a file. See *Save As configuration menu (Measurement Results Table)* on page 143.

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* on page 237.
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 only save 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 multiple bus decode results tables, tap the tab of the bus to configure and then double-tap anywhere in that Bus Decode Results 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.
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)* on page 145.

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* on page 237.
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.

Harmonics Results table configuration menu (Option 5-PWR, SUP5-PWR, or PS2 only)

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

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

***NOTE.** If multiple harmonics results tables are displayed, then tap a harmonics tab to configure and then double-tap in the Harmonics results table area.*

Harmonics 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.
Units panel	
Units	Sets the vertical scale used to display measurement data. Select Log or Linear.
Harmonics	Select **All** to show all harmonics, **Odd** to show just odd harmonics, or **Even** to show just even harmonics in the table. ***NOTE.*** *Changing this field also changes the harmonics bars shown in the Harmonics bar plot.*
Save panel	
Save Table	Opens a menu with which to save the Harmonics Results table data to a comma-separated value format file. See *Save As configuration menu (Harmonics Results Table)*.

Save As configuration menu (Harmonics Results Table)

Use this configuration menu to set the location and file name to save the Harmonics Results table content as a comma-separated value (csv) format file.

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

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

Save As configuration menu (Harmonics Results Table).

Field or control	Description
Save Location	Sets the location 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 the location Use and external keyboard or double-tap on the file name to open the on-screen keyboard and enter the path details. You can also use the Browse button to open a menu to navigate to and select the location to which to save the file. 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 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* on page 237.
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 only save a Harmonics Results table to a comma-separated value (csv) format file.

Annotation and navigation on waveform plots/data and results table (5-PWR, SUP5-PWR, or 5-PS2 only)

Annotation and Navigation are applicable for all the cycle based measurements such as Switching Loss, di/dt, dv/dt, Timing Analysis, and Amplitude Analysis.

To navigate on the results table follow the steps:

1. Tap **Add New... Measurement**.
2. Select the appropriate signals source or sources.
3. Select a supported power measurement (Switching Loss, di/dt, dv/dt, Timing Analysis, or Amplitude Analysis) and tap **Add**.
4. Double-tap the added power measurement badge to open that measurement's configuration menu.
5. Tap **Power Autoset** button for live signals.
6. Select the **Time Trend** Plot if the measurement includes it.
7. Push the **Single/Seq** button on the front panel.
8. Single-tap the measurement badge to display the navigation buttons on the badge.
9. Tap the **Next**/**Previous** buttons to navigate to the next or previous measurement region of the waveform view. The values for each measurement region are displayed above the navigation buttons on the badge. The selected measurement regions are also highlighted in the Zoom view on the Waveform view.
10. Tap the **Min**/**Max** buttons to navigate to the measurement region containing the minimum and maximum measurement values.

Navigation on Bar Graph and Harmonics Results Table (5-PWR, SUP5-PWR, or 5-PS2 only)

1. Add **Harmonics** measurement and configure the parameters.
2. If the source is a live channel, tap **Power Autoset** button.
3. Add Harmonic **Bar Graph**.
4. Tap the Results table and add Harmonics table.
5. Press the **Single/Seq** button, on the front panel.
6. Tap **Harmonics** measurement badge, which expands to display the previous and next button.
7. Tap the navigation button to navigate on the next/previous bar on the Bar Graph and to the respective row in the Harmonic table. This navigation is also vice versa and can be navigated by a single tap on a column or row or using the navigation button.

NOTE.

- *When all the displays are synchronized, Results table row will be highlighted and the selected Bar will be displayed in White color.*
- *If more than one Bar Graph is added, then all the Bar Graphs will be associated with the Results Table at the same time.*

Search configuration menu overview

Use the Search configuration menu to define conditions that you want to mark on a channel or waveform signal.

To open the Search configuration menu, double-tap a Search badge in the results bar.

If there are no Search badges on the Results bar, tap the **Add New... Search** button. A Search badge is added to the Settings bar, and the Search configuration menu opens to search type of Edge (default).

The search types and settings are similar to their corresponding trigger types (Edge, Pulse Width, Runt, and so on).

You can create multiple searches for the same waveform. Each occurrence of the search condition is marked with a triangle along the top of the waveform. Marks for each search are shown in a different color.

Other search types

Bus Search configuration menus on page 148

Edge Search configuration menu on page 164

Logic search configuration menu on page 165

Pulse Width Search configuration menu on page 167

Rise/Fall Time Search configuration menu on page 169

Runt Search configuration menu on page 171

Setup and Hold Search configuration menu on page 172

Timeout Search configuration menu on page 173

Window Search configuration menu on page 175

Bus Search configuration menus

Use a Bus search to search for and mark bus-related events (Start, Stop, Missing Acq, Address, Data, and so on) on a bus waveform.

Select a link to view the configuration menu settings for a specific bus

Bus Search configuration menu, Parallel bus on page 149

Bus Search configuration menu, I2C bus on page 150

Bus Search configuration menu, SPI bus on page 151

Bus Search configuration menu, RS232 bus on page 151

Bus Search configuration menu, CAN bus on page 152

Bus Search configuration menu, LIN bus on page 153

Bus Search configuration menu, FlexRay bus on page 154

Bus Search configuration menu, USB bus on page 156

Bus Search configuration menu, Ethernet bus on page 158

Bus Search configuration menu, MIL-STD-1553 on page 160

Bus Search configuration menu, ARINC 429 on page 162

Bus Search configuration menu, Audio bus on page 163

Bus Search configuration menu, Parallel bus.

Field or control	Description
Source	Select the parallel bus on which to search.
Data	Sets the data pattern on which to search. The number of bits shown depends on how the parallel bus is defined. Tap the Binary or Hex field and use the A and B knobs to select and change the values.
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).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, I^2C bus.

Field or control	Description
Source	Select the I^2C bus on which to search.
Mark When	Select the type of information on which to search.
Direction	Sets the transfer direction on which to search. Only available when Mark When = Address or Address & Data.
Addressing Mode	Sets the slave device address length (7 bits or 10 bits long). Only available when Mark When = Address or Address & Data.
Address	Sets the address pattern on which to search. The number of bits shown depends on the Address Mode setting. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark When = Address or Address & Data.
Data Bytes	Sets the number of data bytes on which to search (one to five bytes). Use the A knob to change the value. Only available when Mark When = Data or Address & Data.
Data	Sets the data pattern on which to search. 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. Only available when Mark When = Data or Address & Data.
A, B knob controls	Use the A knob to select (highlight) the character to change. Use the B knob to change the value of the character.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, SPI bus.

Field or control	Description
Source	Select the SPI bus on which you want to search.
Mark On	Select the type of information on which to search.
Data Words	Sets the number of data words (1 word = 8 bits) on which to search (one to sixteen words). Use the A knob to change the value. Only available when Mark On = Data.
Data	Sets the data pattern on which to search. The number of bits shown depends on the Data Words setting. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark 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).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, RS232 bus.

Field or control	Description
Source	Select the RS232 bus on which to search.
Mark When	Select the type of information on which to search.
Data Bytes	Sets the number of data bytes (1 byte = 8 bits) on which to search (one to ten bytes). Use the A knob to change the value. Only available when Mark When = Data.
Data	Sets the data pattern on which to search. The number of bits shown depends on the Data Words setting. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark When = 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).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, CAN bus.

Field or control	Description
Source	Select the CAN bus on which you want to search.
Mark On	Select the type of information on which to search.
Frame Type	Sets the frame type on which to search. Only available when Mark On = **Type of Frame**.
BRS Bits	Sets the BRS bits state on which to search. Only available when Mark On = **FD Bits**.
ESI Bits	Sets the ESI bits state on which to search. Only available when Mark On = **FD Bits**.
Direction	Sets the transfer direction on which to search. Only available when Mark On = **Identifier**.
Identifier Format	Sets the identifier for Standard (11-bit) or Extended (29-bit for CAN 2.0B) length. Only available when Mark On = **Identifier** or **ID & Data**.
Identifier	Sets the identifier pattern on which to search. The number of bits shown depends on the Identifier Format setting. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = **Identifier** or **ID & Data**.
Data Bytes	Sets the number of data bytes on which to search (one to eight bytes). Use the A knob to change the value. Only available when Mark On = **Data** or **ID & Data**.
Data Offset	Sets the data offset value. Use the A knob to change the value. Only available when Mark On = **Data** or **ID & Data**.
Data	Sets the data pattern on which to search. 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. Only available when Mark On = **Data** or **ID & Data**.
Mark When	Sets the mark when condition. Only available when Mark On = **Data** or **ID & Data**.
A, B knob controls	Use the A knob to select (highlight) the digit to change. Use the B knob to change the value of the digit.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.

Field or control	Description
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, LIN bus.

Field or control	Description
Source	Select to LIN bus on which you want to search.
Mark On	Select the type of information on which to search.
Identifier	Sets the identifier pattern on which to search. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = Identifier or Identifier & Data.
Data Bytes	Sets the number of data bytes on which to search (one to four bytes). Use the A knob to change the value. Only available when Mark On = Data or Identifier & Data.
Data	Sets the data pattern on which to search. 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. Only available when Mark On = Data or Identifier & Data.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a Data Low and Data High boundary pattern for the specified search range. Only available when Mark On = Data or Identifier & 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).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, FlexRay bus.

Field or control	Description
Source	Select the FlexRay bus on which to search.
Mark On	Select the type of information on which to search.
Indicator Bits	Select the defined indicator bits type on which to search from the drop-down list. Only available when Mark On = Indicator Bits.
Indicator Bits	Enter the indicator bits on which to search. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = Header Fields.
Identifier	Enter the frame identifier pattern on which to search. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = Frame ID or Header Fields.
Cycle Count	Enter the cycle count pattern on which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = Cycle Count or Header Fields.
Payload Length	Enter the payload length pattern on which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = Header Fields.
Header CRC	Enter the header CRC pattern on which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = Header Fields.
Data Bytes	Enter the number of data bytes on which to search (one to sixteen bytes). Use the A knob to change the value. Only available when Mark On = Data or Identifier & Data.

Field or control	Description
Data	Enter the data pattern on which to search. The number of bits shown depends on the Data Bytes setting. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = Data or Identifier & Data.
Byte Offset	Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the A knob to change the value. Only available when Mark On = Data or Identifier & Data.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = Frame ID, Cycle Count, Data, or Identifier & Data.
Frame Type	Sets the end of frame type on which to trigger. Only available when Mark On = End of Frame.
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).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, USB bus.

Field or control	Description
Source	Select the USB bus on which to search.
Mark On	Select the type of information on which to search.
Handshake Type	Sets the handshake packet type on which to search. Only available when Mark On = Handshake Packet.
Packet Type	Sets the special packet type on which to search. Only available when Mark On = Special Packet.
Error Type	Sets the error type on which to search. Only available when Mark On = Error.
Address	Sets the token packet address pattern on which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = Token Packet and Token Type = all except SOF (0101).
Token Type	Sets the token packet type on which to search. Only available when Mark On = Token Packet.
Endpoint	Sets the token packet endpoint pattern on which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = Token Packet and Token Type = all except SOF (0101).
Frame Number	Sets the frame number pattern on which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = Token Packet and Token Type = SOF (0101).
Data Packet Type	Sets the data packet type on which to search. Only available when Mark On = Data Packet.
Data Bytes	Sets the number of data bytes on which to search (one to sixteen bytes). Tap the field and use the A knob to change the value. Only available when Mark On = Data Packet.
Byte Offset	Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the A knob to change the value. Only available when Mark On = Data Packet.

Field or control	Description
Data	Sets the data packet pattern on which to search. The number of bits shown depends on the Data Bytes setting. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = Data Packet.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = Token Packet or Data Packet.
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).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, Ethernet bus.

Field or control	Description
Source	Select the Ethernet bus on which to search.
Mark On	Select the type of information on which to search.
MAC Address Destination, MAC Address Source	Sets the MAC destination and/or source address pattern on which to search. Tap a Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = Mac Addresses.
Q-Tag	Sets the Q-tag pattern on which to search. Tap a Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = Q-Tag Control Information.
MAC Length/Type	Enter the MAC length or type pattern on which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = MAC Length/Type
IP Protocol	Sets the IP protocol pattern on which to search. Tap a Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = IP Header.
Source, Destination Address	Sets the source and/or destination IP address pattern on which to search. Tap a Hex or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = IP Header.
Source Port, Destination Port	Sets the source and/or destination TCP header port pattern on which to search. Tap a Binary, Hex or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = TCP Header.
Sequence Number	Sets the TCP header sequence number pattern on which to search. Tap a Binary, Hex or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = TCP Header.
Ack Number	Sets the TCP header ack number pattern on which to search. Tap a Binary, Hex or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = TCP Header.

Field or control	Description
Data Bytes	Sets the number of client data bytes on which to search (one to sixteen bytes). Use the A knob to change the value. Only available when Mark On = Client Data.
Byte Offset	Sets the client data byte offset (Don't Care or the number of bytes). Tap the field and use the A knob to change the value. Only available when Mark On = Client Data.
Client Data	Sets the data pattern on which to search. The number of bits shown depends on the Data Bytes setting. Use in conjunction with the Mark When field to specify the exact trigger condition. Tap the Binary, Hex or ASCII field and use the A and B knobs to select and change the values. Only available when Mark On = Client Data.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Only available when Mark On = MAC Length/Type or Client 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).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, MIL-STD-1553.

Field or control	Description
Source	Select the MIL-STD-1553 bus on which to search.
Mark On	Select the type of information on which to search.
Transmit/Receive Bit	Sets the transmit or receive bit state on which to search. Only available when Mark On = **Command**.
Mark when RT Address	Sets the RT address condition on which to search. When set to **Inside Range** or **Outside Range**, fields are displayed to set a low and high address for the specified search range. Only available when Mark On = **Command**.
Parity	Sets the parity state on which to search. Only available when Mark On = **Command**.
Address	Sets the address value on which to search. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = **Command**.
Low Address	Sets the low address value on which to search. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = **Command** and Mark When RT Address = **Inside Range** or **Outside Range**
Subaddress/Mode	Sets the subaddress or mode value on which to search. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = **Command** and Mark When RT Address = **Inside Range** or **Outside Range**
High Address	Sets the high address value on which to search. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = **Command** and Mark When RT Address = **Inside Range** or **Outside Range**

Field or control	Description
Word Count/Mode Count	Sets the word count or mode count value on which to search. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = **Command** and Mark When RT Address = **Inside Range** or **Outside Range**
Status Word Bits	Sets the status word pattern on which to search. Tap the field and use the A and B knobs to select and change the values. Selecting a bit shows a short description of that bit's function. Only available when Mark On = **Status** and Mark When RT Address = **Inside Range** or **Outside Range**
Data	Sets the data pattern on which to search. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Available when Mark On = **Data**.
Error Type	Sets the error condition on which to search. Available when Trigger On = **Error**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, ARINC 429.

Field or control	Description
Search Type	Set to **Bus**.
Source	Select the **ARINC 429** bus on which to search.
Mark When	Sets the type of information on which to search.
Trigger When Label	Sets the label condition on which to search.
Label	Sets the label pattern on which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Mark When = **Label** or **Label & Data**.
Label Low	Sets the low value of the label pattern range on which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Mark When = **Label** and Mark When Label = **Inside Range** or **Outside Range**.
Label High	Sets the high value of the label pattern range on which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Mark When = **Label** and Mark When Label = **Inside Range** or **Outside Range**.
Trigger When Data	Sets the data condition on which to search.
Data	Sets the data pattern on which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Mark When = **Data** or **Label & Data**.
Data Low	Sets the low value of the data pattern range on which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Mark When = **Data** and Mark When Data = **Inside Range** or **Outside Range**.
Data High	Sets the high value of the data pattern range on which to search. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Mark When = **Data** and Mark When Data = **Inside Range** or **Outside Range**.
SSM	Sets the Sign/Status Matrix (SSM) bit condition on which to search. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Mark When = **Data** or **Label & Data**.
SDI	Sets the Source/Destination Identifier (SDI) bit condition on which to search. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Mark When = **Data** or **Label & Data**.

Field or control	Description
Error Type	Sets the error condition on which to search. Available when Mark When = **Error**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Bus Search configuration menu, Audio bus.

Field or control	Description
Search Type	Set to **Bus**.
Source	Select the Audio bus on which to search.
Mark On	Select the type of information on which to search.
Data	Sets the data pattern on which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark On = Data.
Word	Sets the audio word channel on which to search. Only available when Mark On = Data.
Mark When	Sets the mark when condition for the specified data pattern. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Only available when Mark 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).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types. *Edge Search configuration menu* on page 164

Logic search configuration menu on page 165

Pulse Width Search configuration menu on page 167

Rise/Fall Time Search configuration menu on page 169

Runt Search configuration menu on page 171

Setup and Hold Search configuration menu on page 172

Timeout Search configuration menu on page 173

Window Search configuration menu on page 175

Edge Search configuration menu

Use the Edge search to mark when the specified edge condition occurs on an analog, digital, math, or reference waveform.

Use the Edge Search configuration menu to:

- Create a new Edge search; tap the **Add New... Search** button and set the Search Type to **Edge**.
- Change the settings on an existing Edge search; double-tap an **Edge** Search badge and make necessary changes in the Search configuration menu.

Edge Search configuration menu fields and controls.

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Search Type	Set to **Edge**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
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, falling, or either direction).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types.

Bus Search configuration menus on page 148

Logic search configuration menu on page 165

Pulse Width Search configuration menu on page 167

Rise/Fall Time Search configuration menu on page 169

Runt Search configuration menu on page 171

Setup and Hold Search configuration menu on page 172

Timeout Search configuration menu on page 173

Window Search configuration menu on page 175

Logic search configuration menu

Use the Logic search to mark when specified logic conditions occur on an analog, digital, math, or reference waveform.

To use the Logic Search menu:

- To create a new Logic search, tap the **Add New... Search** button and set the Search Type to **Logic** in the configuration menu.
- To change the settings on an existing Logic search, double-tap a Logic **Search** badge and make necessary changes.

Logic Search configuration menu fields and controls.

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Search Type	Set to **Logic**.
Use Clock Edge?	Enables or disables finding logic patterns that occur on the specified clock edge. Yes places marks on the clock waveform wherever the logic pattern occurs. No places marks on the input signal waveform(s) wherever the logic pattern occurs.
Logic Pattern: Define Inputs	Opens the Logic Search-Define Inputs configuration menu where you define the logic state (High, Low, or Don't Care), and the signal threshold level that defines the logic state for each analog or digital signal. See *Define Inputs*.
Mark When	Defines the waveform logic event to mark, when Use Clock Edge is set to No. ■ **Goes True**: All conditions change to a true state. ■ **Goes False**: All conditions change to a false state. ■ **Is True > Limit**: Condition remains true longer than a specified time. ■ **Is True < Limit**: Condition remains true for less than a specified time. ■ **Is True = Limit**: Condition remains true for a specified time (within ± 5%). ■ **Is True ≠ Limit**: Condition does not remain true for a specified time (within ± 5%).
Clock Source	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform.
Clock Edge	Sets the polarity of the clock edge (rising or falling) for evaluating the other menu conditions. The Logic menu also lets you set the clock edge to either edge.
Clock Threshold	Sets the threshold level that the clock signal must pass through to be considered a valid transition. The clock threshold value is independent of the input signal threshold(s).
Define Logic	Sets the logic condition that must occur with all inputs. ■ **AND**: All conditions are true. ■ **OR**: Any condition is true. ■ **NAND**: One or more conditions are true. ■ **NOR**: No conditions are true.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types.

Bus Search configuration menus on page 148

Edge Search configuration menu on page 164

Pulse Width Search configuration menu on page 167

Rise/Fall Time Search configuration menu on page 169

Runt Search configuration menu on page 171

Setup and Hold Search configuration menu on page 172

Timeout Search configuration menu on page 173

Window Search configuration menu on page 175

Logic Search - Define Inputs configuration menu

Use the Define Inputs menu to select the logic condition to search on, and the logic threshold value, for each channel.

To open the Logic Search-Define Inputs configuration menu:

1. Double-tap a **Logic** Search badge on the Settings bar.
2. Tap the **Logic Pattern > Define Inputs** invoker button.

Logic Search - Define Inputs configuration menu fields and controls.

Field or control	Description
Ch(x) (analog channels) or D(x) (digital Channels	Use to select the signal sources logic condition on which to perform the logic search (**High**, **Low**, **Don't Care**). If a channel is a digital channel, tap the **+** symbol to open the list of digital inputs (D0-D7) from which to select individual logic conditions for the digital signals. To set the threshold levels for digital channels, double-tap the digital Channel badge to open its configuration menu. Use the **Threshold** field to set the signal level that must be exceeded for that signal to be true (logical 1).
Set All	Sets all signal sources to detect a logic **High**, **Low**, or **Don't Care** condition.

Pulse Width Search configuration menu

Use the Pulse Width search to mark a waveform whenever the specified pulse width condition occurs.

To use the Pulse Width Search menu:

- To create a new Pulse Width search, tap the **Add New... Search** button and set the Search Type to **Pulse Width** in the Search configuration menu.
- To change the settings on an existing Pulse Width search, double-tap a **Pulse Width** Search badge and make necessary changes.

Pulse Width Search menu fields and controls.

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Search Type	Set to **Pulse Width**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	■ **< Limit**: A pulse width is less than the specified time limit. ■ **> Limit**: A pulse width is greater than the specified time limit. ■ **= Limit**: A pulse width is equal to the specified time limit (±5%). ■ **≠ Limit**: A pulse width does not equal (is greater than or less than) the specified time limit (±5%). ■ **Inside Range**: A pulse width is in the specified time range (±5%). ■ **Outside Range**: A pulse width is outside of the specified time range (±5%).
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	Sets the longest acceptable pulse width time period for the range condition. Only available when Mark When = Inside Range or Outside Range.
Low Time Limit	Sets the shortest acceptable pulse width time period for the range condition. Only available when Mark When = Inside Range or Outside Range.
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types.

Bus Search configuration menus on page 148

Edge Search configuration menu on page 164

Logic search configuration menu on page 165

Rise/Fall Time Search configuration menu on page 169

Runt Search configuration menu on page 171

Setup and Hold Search configuration menu on page 172

Timeout Search configuration menu on page 173

Window Search configuration menu on page 175

Rise/Fall Time Search configuration menu

Use the Rise/Fall Time search to mark occurrences where the rise or fall time is less than, greater than, equal to, or not equal to a specified time limit.

To use the Rise/Fall Time Search configuration menu:

- To create a new Rise/Fall Time search, tap the **Add New... Search** button and set the Search Type to **Rise/Fall Time** in the Search configuration menu.
- To change the settings on an existing Rise/Fall Time search, double-tap a **Rise/Fall Time** Search badge and make necessary changes.

Rise/Fall Time Search configuration menu fields and controls.

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Search Type	Set to **Rise/Fall Time**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	■ **< Limit**: A signal has a rise time less than the specified time limit. ■ **> Limit**: A signal has a rise time greater than the specified time limit. ■ **= Limit**: A signal has a rise time that is equal to the specified time limit (±5%). ■ **≠ Limit**: A signal has a rise time that does not equal (is greater than or less than) the specified time limit (±5%).
Time Limit	Sets the time period condition to be met.
Slope	Sets the signal transition direction to detect. (rising, falling, or either direction).
Upper Threshold	Sets the upper threshold amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower threshold amplitude level through which the signal must pass to be considered a valid transition.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types.

Bus Search configuration menus on page 148

Edge Search configuration menu on page 164

Logic search configuration menu on page 165

Pulse Width Search configuration menu on page 167

Runt Search configuration menu on page 171

Setup and Hold Search configuration menu on page 172

Timeout Search configuration menu on page 173

Window Search configuration menu on page 175

Runt Search configuration menu

Use the Runt search to mark a waveform where a pulse crosses one threshold but fails to cross a second threshold before recrossing the first threshold.

To use the Runt Search configuration menu:

- To create a new Runt search, tap the **Add New... Search** button and set the Search Type to **Runt** in the Search configuration menu. Use the rest of the menu fields and controls to define the search.
- To change the settings on an existing Runt search, double-tap a **Runt** Search badge and make necessary changes.

Runt Search configuration menu fields and controls.

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Search Type	Set to **Runt**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark 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. ■ **= Limit**: A runt signal event occurs that has a pulse width that is equal to the specified time limit (±5%). ■ **≠ Limit**: A runt signal event occurs that has a pulse width that does not equal (is greater than or less than) the specified time limit (±5%).
Time Limit	Sets the time period condition to be met. Only available when Mark When = < Limit, > Limit, = Limit, or != Limit.
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Upper Threshold	Sets the upper threshold amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower threshold amplitude level through which the signal must pass to be considered a valid transition.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types.

Bus Search configuration menus on page 148

Edge Search configuration menu on page 164

Logic search configuration menu on page 165

Pulse Width Search configuration menu on page 167

Rise/Fall Time Search configuration menu on page 169

Setup and Hold Search configuration menu on page 172

Timeout Search configuration menu on page 173

Window Search configuration menu on page 175

Setup and Hold Search configuration menu

Use the Setup and Hold search type to mark a waveform when a data signal changes state inside of a specified setup and hold time, relative to a specified clock signal.

To use the Setup and Hold Search menu:

- To create a new Setup and Hold search, tap the **Add New... Search** button and set the Search Type to **Setup & Hold** in the Search configuration menu.
- To change the settings on an existing Setup and Hold search, double-tap a **Setup & Hold** Search badge and make necessary changes.

Setup & Hold Search configuration menu fields and controls.

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Search Type	Set to **Setup & Hold**.
Clock Source	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform.
Clock Level	Sets the threshold level that the clock signal must pass through to be considered a valid transition. The clock threshold value is independent of the input signal threshold(s).
Clock Edge	Sets the polarity of the clock edge (rising or falling) for evaluating the other menu conditions. The Logic menu also lets you set the clock edge to either edge.
Data Sources	Sets the data signal source(s). All selected sources must meet the specified setup and hold times. See *Setup and Hold Search - Define Inputs configuration menu* on page 173.
Setup Time	Sets the length of time that data signal should be stable and not change before a clock edge occurs.
Hold Time	Sets the length of time that data signal should be stable and not change after a clock edge occurs.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types.

Bus Search configuration menus on page 148

Edge Search configuration menu on page 164

Logic search configuration menu on page 165

Pulse Width Search configuration menu on page 167

Rise/Fall Time Search configuration menu on page 169

Runt Search configuration menu on page 171

Timeout Search configuration menu on page 173

Window Search configuration menu on page 175

Setup and Hold Search - Define Inputs configuration menu

Use the Define Inputs menu to select the data signal source(s) and set their threshold level(s).

To open the Setup & Hold Search - Define Inputs menu:

1. Double-tap a **Setup & Hold** Search badge on the Results bar.
2. Tap the **Data Sources > Define Inputs** button.

Setup and Hold Search - Define Inputs configuration menu fields and controls.

Field or control	Description
Ch(x) (analog channels) or D(x) (digital Channels	Use to add (**Include**) or exclude (**Don't Include**) the data signal(s) from available input channels and waveforms. If a channel is a digital channel, tap the **+** symbol to open the list of digital inputs (D0-D7) from which to select for that channel. To set the threshold levels for digital channels, double-tap the digital Channel badge to open its configuration menu. For analog channels, use the threshold field to set the data signal level that must be exceeded for the signal transition to be true.
Set All	Use to **Include** or **Don't Include** all available channels and waveforms as data signals.

Timeout Search configuration menu

Use the Timeout search to mark a waveform when it does not detect an expected pulse transition within a specified period of time, such as when a signal gets stuck either high or low.

To use the Timeout Search menu:

- To create a new Timeout search, tap the **Add New... Search** button and set the Search Type to **Timeout** in the Search configuration menu.
- To change the settings on an existing Timeout search, double-tap a **Timeout** Search badge and make necessary changes.

Timeout Search menu fields and controls.

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Search Type	Set to **Timeout**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark 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.
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.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types.

Bus Search configuration menus on page 148

Edge Search configuration menu on page 164

Logic search configuration menu on page 165

Pulse Width Search configuration menu on page 167

Rise/Fall Time Search configuration menu on page 169

Runt Search configuration menu on page 171

Setup and Hold Search configuration menu on page 172

Window Search configuration menu on page 175

Window Search configuration menu

Use the Window search to mark a waveform when the signal rises above an upper threshold level or falls below a lower threshold level (the 'window'), with or without a time limit constraint.

To use the Window Search menu:

- To change the settings on an existing Window search, double-tap a **Window** Search badge and make necessary changes.
- To create a new Window search, tap the **Add New Search** button and set the Search Type to **Window** in the Search configuration menu.

Window Search menu fields and controls.

Field or control	Description
Display	Sets the display of the mark icons on or off. If you have multiple searches defined, the control turns off just the marks for the selected search.
Search Type	Set to **Window**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	■ **Enters Window**: The signal outside a window enters the window defined by the upper and lower threshold settings. ■ **Exits Window**: The signal exits the window defined by the upper and lower threshold settings. ■ **Inside > Limit**: The signal remains inside the window longer then the specified time limit. ■ **Outside > Limit**: The signal remains outside the window longer then the specified time limit.
Upper Threshold	Sets the amplitude threshold value for the upper edge of the window.
Lower Threshold	Sets the amplitude threshold value for the lower edge of the window.
Time Limit	Sets the time period condition to be met. Only available when Mark When = Inside > Limit or Outside > Limit.
Threshold Crossing (Trigger When = Outside > Limit)	■ **Upper**: A signal remains above the upper threshold level for longer than the specified time limit before crossing the upper threshold level to a lower level. ■ **Lower**: A signal remains below the lower threshold level for longer than the specified time limit before crossing the lower threshold level to a higher level. ■ **Either**: A signal remains outside (above or below) the two threshold levels for longer than the specified time limit before crossing either threshold level. ■ **None**: A signal remains outside the two specified threshold levels for longer than a specified time limit.
Threshold Crossing (Trigger When = Inside > Limit)	■ **Upper**: A signal remains between two thresholds for longer than the specified time limit before crossing through the upper threshold. ■ **Lower**: A signal remains between two thresholds for longer than the specified time limit before crossing through the lower threshold. ■ **Either**: A signal remains between two thresholds for longer than the specified time limit before crossing through either the upper or lower threshold. ■ **None**: A signal remains between two threshold levels for longer than a specified time limit.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types.

Bus Search configuration menus on page 148

Edge Search configuration menu on page 164

Logic search configuration menu on page 165

Pulse Width Search configuration menu on page 167

Rise/Fall Time Search configuration menu on page 169

Runt Search configuration menu on page 171

Setup and Hold Search configuration menu on page 172

Timeout Search configuration menu on page 173

Measurement configuration menu overview

Use this configuration menu to add statistics to a measurement badge readout, plot a measurement, and change measurement settings including source, scope (global or local), reference levels, gating, clock recovery, bandwidth filters, and results limits.

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, display plots of the measurement, and so on. 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.
Plots (Measurement name panel)	Buttons that open Plot views of the measurement. Available plots depend on the measurement. Plot types include Time Trend, Histogram, Spectrum, and Eye Diagram (for jitter measurements). To add a plot to the screen, tap the plot button. See *Add Plot configuration menu* on page 141.
Configure panel	Sets the source, label text, and other fields that are specific to each measurement type. See *Configure panel (Measurement configuration menu)* on page 179.
Reference Levels panel	Sets the reference levels and units used to take measurements, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values. See *Reference Levels panel (Measurement configuration menu)* on page 180.
Clock Recovery panel (jitter measurements)	Sets the clock recovery settings for some jitter measurements. See *Clock Recovery panel (Measurement configuration menu)* on page 182.

Field, control, or panel	Description
Gating panel	Sets the measurement region (gate) used to take measurements. Select he scope of the gate setting (global or local), and the type of gating to use. See *Gating panel (Measurement configuration menu)* on page 187.
Filter/Limit Results panel	Sets the scope of the filtering setting (global or local), high and low pass filter settings, the range of measurement result limits, and the limit measurement population size. See *Filter/Limit Results panel (Measurement Settings menu)* on page 188.

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 plots 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.
Plots	Adds a Plot view of the measurement value to the screen. Available plots depend on the measurement. Plot types include Time Trend, Histogram, Spectrum, and Eye Diagram (for jitter measurements). Trend adds the trend plot to the Waveform view.

See also.

Configure panel (Measurement configuration menu) on page 179

Reference Levels panel (Measurement configuration menu) on page 180

Clock Recovery panel (Measurement configuration menu) on page 182

Gating panel (Measurement configuration menu) on page 187

Filter/Limit Results panel (Measurement Settings menu) on page 188

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.
Signal Type	Sets the signal type (Clock, Data, Auto) of the source signal for some measurements.
Edge, Clock edge	Sets the edge of the signal to use for starting the measurement.
From Edge	Sets the Source 1 waveform edge on which to start the measurement, for two-source measurements.
To Edge	Sets the Source 2 waveform edge on which to stop the measurement, for two-source measurements.
Calculate One Measurement Per	Sets the amount of waveform data to use to calculate one measurement; one measurement across the entire waveform record or one measurement for each cycle of the waveform in the record.
Pattern Detection	Auto attempts to detect the type of pattern and set
Pattern Type	Sets whether the source signal data is a Repeating pattern or an Arbitrary pattern type.
Pattern Length	Sets the pattern length to use when Pattern Type = Repeating.
Window Length	Sets the overall window length to use when Pattern Type = Arbitrary.

Other measurement panels.

Measurement Name panel (Measurement configuration menu) on page 178

Reference Levels panel (Measurement configuration menu) on page 180

Clock Recovery panel (Measurement configuration menu) on page 182

Clock Recovery- Advanced Settings configuration menu on page 186

Gating panel (Measurement configuration menu) on page 187

Filter/Limit Results panel (Measurement Settings menu) on page 188

Reference Levels panel (Measurement configuration menu)

Use the Reference Levels panel to set the scope of the reference level settings (global or local), 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.

Field or control	Description
Reference Levels	Global sets whether the reference levels defined in this measurement apply to all measurements that are set to global (the default setting). Local sets the Reference Level parameters to apply to just this measurement.
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%.
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, and automatically determines the best Base Top method to use. Most commonly sets the Top Base method to Histogram Mode. **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 Mean** uses histogram analysis to calculate the mean or average value using all values above and below the waveform midpoint. Top is set to the mean high value, and Base is set to the mean low value. **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. **Histogram Eye Center** uses histogram analysis to determine the base top amplitude. Creates a histogram of the amplitudes in the center of each bit (unit interval) while ignoring the waveform during bit transitions. The histogram should have a peak at the nominal high level and another peak at the nominal low level. ***NOTE.*** *If you set the Base Top Method to other than Auto, and do not change the Reference Levels mode to Local, many existing measurements, as well as measurements that you add to the Results bar, will use the new Base Top Method values for taking measurements. This may result in measurement values that you are not expecting.*

Field or control	Description
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) on page 178

Configure panel (Measurement configuration menu) on page 179

Clock Recovery panel (Measurement configuration menu) on page 182

Clock Recovery- Advanced Settings configuration menu on page 186

Gating panel (Measurement configuration menu) on page 187

Filter/Limit Results panel (Measurement Settings menu) on page 188

Clock Recovery panel (Measurement configuration menu)

Clock recovery refers to the process of establishing a reference clock, the edges of which can be used as a basis for timing comparisons. Use the Clock Recovery panel to configure the clock recovery settings for measurements that require a clock signal.

To access the Clock Recovery panel:

1. Double-tap a Measurement badge on the Results bar that uses clock recovery (such as jitter measurements), to open the Meas configuration menu.
2. Tap the **Clock Recovery** panel.

Measurement configuration menu, Clock Recovery panel.

Field or control	Description
Clock Recovery	**Global** sets whether the Clock Recovery settings defined in this panel apply to all measurements with clock recovery settings that are set to global (the default setting). **Local** sets the Clock Recovery parameters to apply to just this measurement.
Method	**PLL** simulates the behavior of the specified hardware Phase Locked Loop clock recovery circuit to derive the clock signal. **Constant Clock** uses linear regression so that the recovered clock minimizes the mean squared sum of the Time Interval Error (TIE) for that waveform. **Explicit Clock** derives the reference clock from a specified channel other than the one upon which the measurement is defined.
Standard	Sets the standard to use for the PLL model. Information for a selected standard is listed under the drop-down list. Only available when Method = PLL.
Mode	**Mean** chooses both the frequency and the phase to minimize the mean squared error. Available when Method = Constant Clock. **Median** chooses the phase so that the median error between the recovered and measured edges is zero. Only available when Method = Constant Clock. **Fixed** uses the specified frequency but chooses the phase so that the median error between the recovered and measured edges is zero. Only available when Method = Constant Clock. **Select Explicit Clock-Edge** method if you want to use the edges found in the selected clock source (possibly multiplied up by an integral number). If the Clock Multiplier is set to 1 (the default), only these edges will be used. If the Clock Multiplier is set to a number N other than 1, linear interpolation will be used between each pair of actual edges to create N-1 additional reference edges. The interpolated edge times, combined with the actual edges, give a total of N reference edge times per actual edge. Only available when Method = Explicit Clock. **Select Explicit Clock-PLL** as the clock recovery method if you want to feed the edges from the selected clock source through a PLL rather than using them directly. The actual edges from the clock source will be used to drive a software PLL model, and the edge times coming out of the PLL will be used as the reference edges for the target measurement. If the Clock Multiplier is set to a number N other than 1, the output of the PLL will have N edges per actual edge. Only available when Method = Explicit Clock.

Field or control	Description
Calculate On	**First Acq.** sets the clock-recovery algorithm to choose a new best-fit clock frequency and phase on just the first acquisition. Subsequent acquisitions will choose a best fit on clock phase but retain the clock frequency found in the first acquisition. **Every Acq.** the clock-recovery algorithm to choose a new best-fit clock frequency and phase for each new oscilloscope acquisition. Clearing the measurement results will reset the clock recovery so that both frequency and phase are optimized on the subsequent acquisition. Only available when Method = Constant Clock and Mode = Mean or Median.
Clock Frequency	Sets the clock frequency to use in Fixed mode. The clock in the waveform is ignored; the instrument uses the specified frequency, with the clock phase determined by best fit. ***NOTE.*** *This method typically results in a closed eye.* Only available when Method = Constant Clock and Mode = Fixed.
Loop BW	Sets the PLL loop bandwidth. Displays the Closed Loop bandwidth that has been configured based on the current standard. Only available when Method = PLL.
Clock Source	Sets the source for the explicit clock. Only available when Method = Explicit Clock.
Clock Edge	Set whether the rising, falling or both edges of the clock source should be considered. Only available when Method = Explicit Clock.
Clock Multiplier	Set the number of edges to be used. If the Clock Multiplier is set to 1 (the default), only these edges will be used. If the Clock Multiplier is set to a number N other than 1, linear interpolation will be used between each pair of actual edges to create N-1 additional reference edges. The interpolated edge times, combined with the actual edges, give a total of N reference edge times per actual edge. Only available when Method = Explicit Clock.
Clock Offset	Set to Auto or Manual.
Offset	Sets the clock offset amount relative to data. To compare the reference clock times to the edge times from the data source, some assumptions must be made about how they align. The default assumption is that each data source edge is associated with the reference clock edge to which it is nearest in time. This assumption may not be optimum, for example if the probes for the reference clock and data signal have different cable lengths. Only available when Clock Offset = Manual.

Field or control	Description
PLL Model	Select the PLL model type. The PLL control area provides control over the phase-locked loop used for clock recovery. You can choose the loop bandwidth and the loop order, and if a Type II loop is chosen, you can specify the damping factor. To set the loop bandwidth automatically, based on a serial standard, select PLL: Standard BW as the clock recovery method. From the Standard: b/s list box, select the standard that matches your data link. For example, choose “PCI-E: 2.5” to test a 2.5 Gbit/second PCI Express link. In this case, the PLL bandwidth will be set to 1.5 MHz, which is 1/1667 of the baud rate as specified in PCI Express standard. You can use the PLL Model list box to choose between Type I and Type II loop. A Type I loop has a transfer function that approaches zero frequency with a slope of 1/s and a Type II loop approaches zero frequency with a 1/s2 slope (In much of the PLL literature, these terms are used interchangeably with First-Order and Second-Order loops. For a thorough discussion of loop type versus order, see Frequency Synthesis by Phase Lock, by William Egan). Only available when Method = Explicit Clock and Mode = Explicit Clock - PLL.
JTF BW	Displays or sets the Jitter Transfer Function bandwidth that has been configured based on the current standard. Only available when Method = Explicit Clock and Mode = Explicit Clock - PLL and PLL Model = Type II.
Damping	Sets the damping factor for the PLL. It is enabled only for Type II phase-locked loop. Only available when Method = Explicit Clock and Mode = Explicit Clock - PLL and PLL Model = Type II.
Advanced	Opens the Clock Recovery-Advanced Settings configuration menu to refine the recovered clock signal. See *Clock Recovery- Advanced Settings configuration menu* on page 186. Only available when Method = PLL.

About constant clock recovery. In Constant Clock Recovery, the clock is assumed to be of the form A*sin (2π ft +phase), where the frequency (f) and phase are treated as unknown constants. Once a source waveform has been acquired and the edges extracted, one or both of these constants are determined using linear regression, so that the recovered clock minimizes the mean squared sum of the Time Interval Error (TIE) for that waveform.

About PLL loop BW versus JTF BW. Phase locked loops are characterized according to their bandwidth (BW), and several different bandwidths are commonly used. The terminology used for these bandwidths is described here, since it varies somewhat across different industries.

- Loop BW (or Closed Loop BW) is the frequency at which the closed-loop gain has fallen to -3 dB (half power) relative to unity-gain. The closed-loop gain function has the character of a low-pass filter.
- JTF BW (Jitter Transfer Function BW or Error Function BW) is the frequency below which input jitter to a tracking loop is removed. The JTF BW has a high-pass filter characteristic.

For Type I loops, the Loop BW and the JTF BW are always equal. For Type II loops, these two bandwidths are different, and their ratio depends on the PLL damping factor. You can choose to specify either bandwidth, and the other is displayed for reference.

PLL-based clock recovery. PLL-based clock recovery is implemented using a software model of a hardware PLL circuit, sequentially processing waveform transitions and adjusting the clock period in a feedback loop. This approach means that the transition density of the input signal has subtle effects on the effective bandwidth and damping factor of the feedback loop, just as it does with actual hardware PLLs. The influence of transition density is only relevant for data signals, since clock signals (or data signals with a two bit pattern) have 100% transition density.

Other measurement settings panels.

Measurement Name panel (Measurement configuration menu) on page 178

Configure panel (Measurement configuration menu) on page 179

Reference Levels panel (Measurement configuration menu) on page 180

Clock Recovery- Advanced Settings configuration menu on page 186

Gating panel (Measurement configuration menu) on page 187

Filter/Limit Results panel (Measurement Settings menu) on page 188

Clock Recovery- Advanced Settings configuration menu

Sets advanced clock recovery settings such as defining a nominal data rate or basing clock recovery on a known data pattern.

To open the Clock Recovery-Advanced Settings configuration menu:

1. Double-tap a measurement that requires clock recovery information (such as jitter measurements).
2. Tap the **Clock Recovery** panel
3. Tap the **Advanced** button.

Clock Recovery-Advanced Settings configuration menu, fields and controls.

Field or control	Description
Advanced Clock Recovery Method	Sets the method used to recover a clock from the measured signal. Available selections are **None** (default), **Nominal Data Rate** and **Known Data Pattern**.
Bit Rate	Sets the clock bit rate. The **Bit Rate** field is only present when **Advanced Clock Recovery Method** is set to **Nominal Data Rate**.
Pattern File	Drop-down list with 20 most recent pattern files loaded onto the oscilloscope. Available when Advanced Clock Recovery = Known Data Pattern.
Browse	Opens a standard file navigation window. Use to navigate to and select a pattern file.

See also.

Measurement Name panel (Measurement configuration menu) on page 178

Configure panel (Measurement configuration menu) on page 179

Reference Levels panel (Measurement configuration menu) on page 180

Clock Recovery panel (Measurement configuration menu) on page 182

Gating panel (Measurement configuration menu) on page 187

Filter/Limit Results panel (Measurement Settings menu) on page 188

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.

Field or control	Description
Gating	Sets whether this measurements gating settings are **Global** or **Local**. When **Global** is selected, changing anything in this panel causes the same change to all other measurements that also have **Global** selected. When **Local** is selected, settings in this panel only effect this measurement.
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. **Logic**: Measurements are taken only when the logical state of a specified waveform is true.
Source	Sets the source to use for Logic or Search gates. If Gating Type = **Logic**, the Source field lists all analog channels, math waveforms and reference waveforms. If Gating Type = **Search**, the Source field lists all defined searches.
Threshold	Sets the threshold value for the Logic gate source to be considered a logic 1 value.
Hysteresis	Sets the Hysteresis value for the Logic gate source.
Active	Sets the logic state value for the Logic gate source.

See also.

Measurement Name panel (Measurement configuration menu) on page 178

Configure panel (Measurement configuration menu) on page 179

Reference Levels panel (Measurement configuration menu) on page 180

Clock Recovery panel (Measurement configuration menu) on page 182

Clock Recovery- Advanced Settings configuration menu on page 186

Filter/Limit Results panel (Measurement Settings menu) on page 188

Filter/Limit Results panel (Measurement Settings menu)

Use these settings to apply a High Pass and/or Low Pass filter to block specified frequency band components when taking measurements. Use the limit controls to set range of measurement values to measure, and the number of measurements to take (population).

To open the **Filter/Limit Results** panel:

1. Double-tap a Measurement badge.
2. Tap the **Filter/Limit Results** panel.

Filter/Limit Results panel fields and controls. Not all items listed in the table may be shown for all measurements; The panel only shows fields and controls relevant to the selected measurement.

Field or control	
Filter Measurement Results	Sets whether this measurement's filter and limits settings are **Global** or Local. When set to **Global** (the default), changing anything in this panel causes the same change to all other measurement filter and limit settings that also have Global selected. When set to **Local**, changing anything in this panel only effects this measurement.
High Pass Filter	Blocks the low frequency band and passes only the high frequency band of the waveform. Select a Butterworth filter order (No Filter (default), 1st, 2nd, or 3rd) and enter the roll-off frequency in the field.
FP Freq (F1)	High Pass filter cut-off frequency at which the filter magnitude falls by 3 dB.
Low Pass Filter	Blocks the high frequency band and passes only the low frequency band of the waveform. Select a Butterworth filter order (No Filter (default), 1st, 2nd, or 3rd) and enter the roll-off frequency in the field.
LP Freq (F2)	Low Pass filter cut-off frequency at which the filter magnitude falls by 3 dB.
Limit Measurement Results	Limit taking measurements results to those that are within the specified Min Value and Max value range.
Limit Measurement Population	Limit measurements to the specified number of measurements.

Other measurement settings panels.

Measurement Name panel (Measurement configuration menu) on page 178

Configure panel (Measurement configuration menu) on page 179

Reference Levels panel (Measurement configuration menu) on page 180

Clock Recovery panel (Measurement configuration menu) on page 182

Clock Recovery- Advanced Settings configuration menu on page 186

Gating panel (Measurement configuration menu) on page 187

Power measurement configuration menu overview (option 5-PWR, SUP5-PWR, or 5-PS2 only)

Use this configuration menu to add statistics to a Power measurement badge readout, plot a measurement, and change measurement settings including source, scope (global or local), reference levels, and gating.

To open the Power measurement configuration menu for a measurement, double-tap a Power 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, display plots of the measurement, and so on. The measurement name panel only shows fields and controls relevant to the selected measurement. The most common Power measurement Name fields are listed in the following table.

Power 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 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.
Plots (Measurement name panel)	Buttons that open Plot views of the measurement. Available plots depend on the measurement. Plot types include Time Trend, Histogram, Harmonic bar graph, SOA, Switching Loss Trajectory, and Instantaneous Math. To add a plot to the screen, tap the plot button. See *Add Plot configuration menu* on page 141.
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, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values. See *Reference Levels panel (Measurement configuration menu)* on page 180
Gating panel	Sets the measurement region (gate) used to take measurements. Select the scope of the gate setting (global or local), and the type of gating to use. See *Gating panel (Measurement configuration menu)* on page 187

Power Measurement Name panel (Measurement configuration menu)

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

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

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

Field or control	Description
Power Autoset	Sets the oscilloscope acquisition system for optimal measurement results for all active power measurements. See *Power Autoset*.
Measurement Statistics	Shows 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	Adds the listed statistical measurement readouts to the measurement badge.
Plots	Adds a Plot view of the measurement value to the screen. Available plots depend on the measurement. Plot types include SWL Trajectory, Bar Graph, Time Trend, Histogram, and SOA. Trend adds the trend plot to the Waveform view.

See also.

Configure panel (Measurement configuration menu) on page 179

Reference Levels panel (Measurement configuration menu) on page 180

Gating panel (Measurement configuration menu) on page 187

Configure panel (Power measurement configuration menu)

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

To open the Configure panel for a power measurement:

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

di/dt, dv/dt measurement: Configure panel.

Field or control	Description
Source	Sets the signal source used to take the measurement. Tap the field to show the list of available sources. Select the current source.
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.
Edge	Sets the signal edges to detect (rise or fall).

Harmonics measurement: Configure panel .

Field or control	Description
Standard	Set the standard to use for measurements. None (no Standard), IEC 61000-3-2, MIL-STD-1399, or AM14.
Harmonics	Set the harmonics order (number of harmonics) for the selected standard. Ranges from 40 to 100.
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
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.
Harmonics Source	Selects the source of harmonics calculation source (Voltage or Current) for computation of the Harmonics.
Power Level (MIL-STD-1399)	Select the required power level, High or Low. It is used to compute limit values for MIL-STD-1399.
Current (MIL-STD-1399)	Select either rated or measured. Specify the value for rated or measured the Input current signal.

Power Quality measurement: Configure panel.

Field or control	Description
Voltage Source	Configures the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Configures the current source used to take the measurement. Tap the field to show the list of available sources.
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.
Calculate Over Full Cycles	Allow to calculate over the entire acquisition.
Frequency Reference	Select the signal source (Voltage or Current) to use to determine the measurement frequency.

SOA measurement: Configure panel.

Field or control	Description
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
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.
Define Mask	Defines the linear mask for SOA measurements. See *Define Mask*

Switching Loss measurement: Configure panel.

Field or control	Description
Voltage Source	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
Current Source	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
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.
Type	Select SMPS/ PFC/Flyback: ■ SMPS: Select this option in case of signals without noise and ringing. The Vg source is not required. Select Vg source (Source 3), in case of noisy signal. ■ PFC: Select this option when input DUT signals are from Power Factor Correction Circuit. For this case, Vg source is mandatory. ■ Flyback: Select this option when input signals are ringing. This option does not require a Vg source. ***NOTE.*** *Displayed REF levels shown in the Reference Levels panel after selecting Flyback are default values and not the internally computed REF values used for measurement calculations.*
Gate Voltage (Vg)	Sets the Vg input source, which is a clean signal. Available when Type = **SMPS** or **PFC.**
Vg Level Ton - Start	Allows to select the source of computation of the harmonic. In the voltage source, standard is always **None**. Available when Gate Voltage (Vg) **≠None**.
PWM Type	Select **Fixed** or **Variable** based on the varying pulse width of the switching signal.
Conduction calculation	Select **MOSFET** or **BJT/IGBT** semiconductor types.
R_{DS}(on)	If MOSFET is selected, then R_{DS}(on) is used to compute for conduction Loss. Available when Conduction Calculation = **MOSFET**
V_{CE}(sat)	If DJT/IGBT is selected, then V_{CE}(sat) is used to compute for conduction loss. Available when Conduction Calculation = **BJT/IGBT**
Set On/Off Levels In:	Sets the REF levels for computation of the T_{ON} an T_{OFF} regions. Levels can be set in % or absolute values.
T_{off}-Stop Current Level	Used to set T_{on}-Start and T_{off}-Stop of the max switch current.
T_{on} - Stop & T_{off}- Start Voltage Level	Sets the voltage level value for T_{off}-Stop and T_{on}. Can be entered as a percent or as a voltage, depending on the setting of the **Set On/Off Levels In:** control.

Amplitude Analysis, Timing Analysis power measurements: Configure panel.

Field or control	Description
Source	Sets the signal source or sources used to take the measurement. If the measurement requires more than one source, multiple source fields are displayed. Tap the field to show the list of available sources.
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.
Edge	Sets the signal edge to detect (rise or fall). Used in most **Timing Analysis** measurement. Available when the power measurement = **Frequency**.
Clock Edge	Sets the clock signal edge to detect (rise, fall, or either). Used in **Timing Analysis** measurements. Available when the power measurement = **Positive Duty Cycle** or **Negative Duty Cycle**.
Line Frequency	Sets the line frequency of the power signal source. **Power Autoset** uses the selected frequency to setup the scope acquisition parameters. Available when the power measurement = **Line Ripple**.
Switching Frequency	Specify the operating frequency of the switching device. **Power Autoset** uses the selected frequency to set the oscilloscope acquisition parameters. Frequency range is 50 Hz to 1 MHz. Only available when the power measurement = **Switching Ripple**.

Other measurement configuration panels .

Reference Levels panel (Measurement configuration menu) on page 180

Power Measurement Name panel (Measurement configuration menu) on page 190

Power Measurement Name panel (Measurement configuration menu) on page 190

Gating Panel (Measurement configuration menu)

Gating panel (Power measurement configuration menu) on page 198

Power Autoset

Sets the oscilloscope acquisition system for optimal measurement results for all active power measurements.

Before you tap the Power Autoset button, follow the steps below:

1. Configure the appropriate input source for the added measurements.
2. Connect the appropriate input signals for the added measurements.

3. Configure all the added power measurements by configuring all the measurements individually.
4. Tap **Power Autoset** button and wait for the busy indicator to disappear.

NOTE.

- *In case of failure, a time bound message appears in a popup message. See Errors and Warnings.*
- *When different frequency signals are connected to two different channels, Power Autoset uses the first AC signal connected to compute and setup the scope parameters. For example, If channel 1 is connected to DC signal and channel 2 to AC signal of 1 MHz, and channel 6 to AC signal of 1 KHz. Power Autoset considers channel 2 (the first AC signal), for setting up the scope parameters. Channel 6 may not be set up to see the minimum number of cycles properly. The user should setup the scope parameter for channel 6 manually.*

SOA Mask definition controls and fields

Use the SOA Mask dialog to configure the parameters to add point, delete point, save mask, and recall mask.

Use the parameters to define the linear mask for an SOA measurement.

Define Mask fields and controls

Field or control	Description
X (Volts)	Define voltage values for the mask point.
Y (Amps)	Define current values for the mask point.
Add Point	Add voltage and current points to define mask. Points are added to the end of the existing list.
Delete Point	Deletes the selected point data row.
Save Mask	Opens the Save As menu to navigate to and select the location at which to save the SOA mask data as a .pwrmsk file.
Recall Mask	Opens the Open menu to navigate to and select the location from which to recall (load) the SOA mask data .pwrmsk file.

Save Mask menu (SOA power measurement)

Use this menu to save a SOA power measurement mask file to a specified location.

Prerequisite: Open the **Configure** panel of the SOA measurement for which you want to save a mask file (.pwrmsk).

To open the Save Mask configuration menu:

1. Tap the **Save Mask** button to open the **Save As** menu.
2. Use the menu fields and controls to navigate to and select the location to save the SOA mask file.

Save As configuration menu (SOA mask file).

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. Use the **Browse** button to quickly navigate to the location to which to save the file. Or you can 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 virtual 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.
Browse	Tap to open the Browse Save As Location configuration menu, to navigate to the location to which to save the file. See *Browse Save As Location configuration menu* on page 237 .
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 the default value of Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. 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 SOA mask files as type .pwrmsk.
Cancel	Cancels the file save action and closes the configuration menu. You can also cancel the save operation by tapping anywhere outside the menu.
Save	Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.

Recall Mask menu (SOA power measurement)

Use this menu to recall (load) a .pwrmsk mask file for an SOA power measurement plot.

Prerequisite: Open the **Configure** panel of the SOA measurement for which you want to recall a mask (pwrmsk) file.

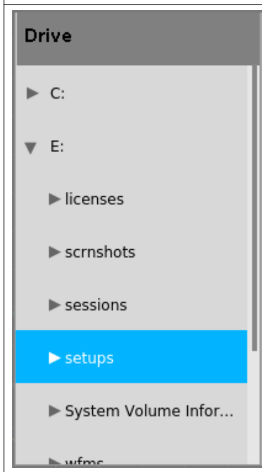
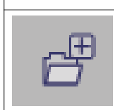
To open the Recall Mask configuration menu:

1. Tap the **Recall Mask** button to open the **Recall** menu.
2. Use the menu fields and controls to navigate to and select the mask file to recall.

File operations and Option 5-WIN (Microsoft Windows 10 Operating System SSD) 5 Series MSO only (not MSO58LP). Instruments with Option 5-WIN installed (Windows 10 SSD) will display the standard Windows file tools to navigate to and select files and folders.

The Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. This is different from standard instruments (without 5-WIN installed), which assign a non-changing drive letter to each USB port.

Recall file configuration menu (SOA mask file).

Field or control	Description
Look in:	Shows the current directory path to the location of a file. 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 virtual 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, up to a maximum of 20 locations.
	The Drive column lists the directory structure, opening at the root (/) level. Use to quickly navigate to a location. Tap to list the contents of the directory in the Name pane. Double-tap an item to display the directory and any subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
	Use the arrow buttons to navigate the file directory. 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.
	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 to recall. Tap on the file name in the Name column to add it to this field.
Files of type	The SOA mask file type (.pwrmsk) cannot be changed.
Cancel	Cancels the file recall action and closes the menu.
Recall	Recalls the selected SOA mask file and plots it on the SOA Plot view.

USB port drive names and locations. Use the table to determine which drive to select when navigating to and/or selecting a file on system memory or a connected USB memory device.

Drive name	Drive letter	Drive or physical USB port location
MSO54, MSO56, MSO58		
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**E**	USB 3.0 (left)
	F	USB 2.0 (center)
	G	USB 2.0 (right)
Rear panel	**H**	USB 2.0 (top)
	I	USB 2.0 (bottom)
	J	USB 3.0 (top)
	K	USB 3.0 (bottom)
MSO58LP		
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**E**	USB 3.0 (left)
	F	USB 2.0 (right)
Rear panel	**G**	USB 2.0 (top)
	H	USB 2.0 (bottom)
	I	USB 3.0 (top)
	J	USB 3.0 (bottom)
Option 5-WIN, SUP5-WIN instruments and USB port labels		
Root drive	**C**	User-accessible memory on the oscilloscope.
USB ports	**Dynamic port letter assignment**	If option 5-WIN or SUP5-WIN is installed (Windows operating system), the Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. Use standard Windows procedures to mount and access network drives.

Reference Levels panel (Power measurement configuration Menu)

Use the Reference Levels panel to set the scope of the reference level settings (global or local), the measurement 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 reference levels to be the same or different for rising and falling edges.

See *Reference Levels panel (Measurement configuration menu)* on page 180 for the Reference panel fields and controls.

Other measurement settings panels. *Power Measurement Name panel (Measurement configuration menu)* on page 190

Configure panel (Power measurement configuration menu) on page 190

Gating panel (Power measurement configuration menu) on page 198

Gating panel (Power measurement configuration menu)

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

See *Gating panel (Power measurement configuration menu)* on page 198 for the Gating panel fields and controls.

See also

Power Measurement Name panel (Measurement configuration menu) on page 190

Reference Levels panel (Power measurement configuration Menu) on page 198

Configure panel (Power measurement configuration menu) on page 190

Analog Channel configuration menu

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

To open an analog Channel configuration menu, double-tap an analog Channel badge. The following text describes analog channel settings. For digital channel settings, see *Digital channel configuration menu* on page 229.

Vertical Settings panel, fields and controls

Field or control	Description
Display	Tap to toggle display of the channel On and Off.
Vertical Scale	Double-tap the field to set the scale using the multipurpose knob, a virtual keypad, or tap the up or down arrows to change the scale.
Offset	Double-tap the field to set the offset using the virtual keypad.
Set to 0	Tap to set the offset to 0.
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 slice or waveform view.
Label	Double-tap the field to add a label to the channel display using the virtual keypad.
Coupling	Tap to set the input coupling to DC or AC.
Termination	Tap to set the input termination to 1 MΩ or 50 Ω. If you are using a supported TekVPI probe, this value is automatically set by the probe and these controls are not available.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.

Field or control	Description
Other settings panels	
Probe Setup	Use to see probe information, check the probe compensation status, compensate the probe, or restore the factory defaults. See *Probe Setup panel (Channel configuration menu)* on page 199. See *Probe Compensation configuration menu (analog channels Probe Setup panel)* on page 200. See *Unsupported Probe dialog* on page 200.
Other	Use to adjust signal delay to align signal arrival at the oscilloscope between probes and/or cables. See *Other panel (Channel configuration menu)* on page 201.

Probe Setup panel (Channel configuration menu)

Use the Channel configuration menu Probe Setup panel to see probe information, check the probe compensation status, compensate the probe, or restore the factory defaults.

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 Information	View probe information such as probe type, serial number, and its attenuation.
Probe Compensation Status	View the probe compensation status: Default, Pass, or Fail.
Compensate Probe	Tap to display the probe compensation dialog. This is only available for probes that support automatic compensation.

See also.

Unsupported Probe dialog on page 200

Probe Compensation configuration menu (analog channels Probe Setup panel) on page 200

Other panel (Channel configuration menu) on page 201

Deskew configuration menu (Other panel, Channel configuration menu) on page 201

Unsupported Probe dialog

This dialog tells you that you have attached an unsupported probe.

To open the Unsupported Probe dialog:

1. Double-tap a Channel badge to open the channel configuration menu.
2. Tap the **Probe Setup** panel.
3. Read the Probe Information.

If you have attached an unsupported probe, remove the unsupported probe and attach a supported probe to the input channel.

See also.

Unsupported Probe dialog on page 200

Probe Compensation configuration menu (analog channels Probe Setup panel) on page 200

Other panel (Channel configuration menu) on page 201

Deskew configuration menu (Other panel, Channel configuration menu) on page 201

Probe Compensation configuration menu (analog channels Probe Setup panel)

Use this menu to compensate probes that support automatic frequency compensation. This menu is only available when a compensation-supported probe is installed on the channel.

To open the Probe Compensation dialog:

1. Double-tap the Channel badge on the Settings bar to open the channel configuration menu.
2. Tap the **Probe Setup** panel.
3. Tap **Compensate Probe**.

Probe Compensation dialog. Available fields and controls vary with the type of probe that is attached. For more information, consult the probe documentation. Read the information on the menu before starting the probe compensation process.

Field or control	Description
Compensate Probe	Tap to compensate the attached probe. Before compensating the probe, read the instructions in the dialog.
Restore Factory Defaults	Tap to restore the probe compensation factory defaults and remove the previous compensation results.
Probe Compensation Status	Probe compensation status can be Running, Passed, Failed or Default.
OK	Tap to close the dialog.

See also.

Unsupported Probe dialog on page 200

Probe Compensation configuration menu (analog channels Probe Setup panel) on page 200

Other panel (Channel configuration menu) on page 201

Deskew configuration menu (Other panel, Channel configuration menu) on page 201

Other panel (Channel configuration menu)

Use the Other panel to set the channel deskew, external attenuation, and alternate vertical scale units.

To open the analog Channel configuration menu Other panel:

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

Other panel fields and controls.

Field or control	Description
Deskew	Sets or displays the probe deskew value.
Set to 0	Sets the probe deskew value to zero (0) seconds.
Multi-Channel	Opens a Deskew configuration menu that allows you to deskew multiple channels (two at a time).
External Attenuation	Double-tap the numeric fields to set the external attenuation using the virtual keypad.
Set to Unity	Sets the external attenuation to unity.
Alternate Units	Toggles Alternate Units On or Off. Use to set custom vertical measurement units. The vertical scale will show the entered measurement units.
Units	Double-tap to enter alternate units using the virtual keyboard. Only shown when Alternate Units = On.
Ratio	Double-tap the numeric fields to set the ratio using the virtual keypad. Only shown when Alternate Units = On.
Set to Unity	Sets the ratio to unity. Only shown when Alternate Units = On.

See also.

Unsupported Probe dialog on page 200

Probe Compensation configuration menu (analog channels Probe Setup panel) on page 200

Other panel (Channel configuration menu) on page 201

Deskew configuration menu (Other panel, Channel configuration menu) on page 201

Deskew configuration menu (Other panel, Channel configuration menu)

Use the Deskew configuration menu to make display and measurement adjustments for analog probes that have differing propagation delays. This is especially important when using a current probe in conjunction with a voltage probe.

To open the Deskew configuration menu:

1. Double-tap an analog Channel badge on the Settings bar to open the Channel configuration menu.
2. Tap the **Other** panel.
3. Tap the **Multi-Channel** button.

Use the controls in the Deskew menu to set the deskew parameters to recommended values, based on the nominal propagation delay of supported probes. The oscilloscope automatically loads the nominal propagation delay values of TekVPI and TekProbe II probes (TekProbe II probes require use of a TPA-BNC adaptor).

***NOTE.** This deskew menu does not actively test and adjust the probe delay between channels; it uses the delay values stored in supported probes, or a custom propagation delay value that you enter, to set the propagation delay to zero between the reference channel probe and one or more other probes.*

To actively adjust probe delay using a signal, see Deskew analog input channels - quick visual method on page 63 and Deskew analog input channels - measurement method on page 64.

Deskew menu 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
From Source	Tap and select from the drop-down list the channel to deskew from (your reference channel for deskewing).
To Source	Tap and select from the drop-down list the channel to deskew to (the channel that you want to match the From Source reference channel).
Probe	If the oscilloscope recognizes the probe attached to the channel, the Probe field shows the nomenclature of the attached probe. If the oscilloscope does not recognize the probe attached to the channel, the Probe field shows a drop-down list from which you can select the probe that is attached to the selected channel. If the attached probe is not in the list, select Custom (at the bottom of the list) and enter the probe propagation delay in the Prop. Delay field.
Prop. Delay	Field in which you can enter the probe delay for an unsupported probe. Double-tap in the field and use the keypad to enter the propagation delay value. This field is available when Probe = Custom.
Propagation Delay	This field lists the default propagation delay of the attached probe. A positive value shifts a channel to the left.
OK, deskew	Sets the oscilloscope to add or subtract the delay values of the To Source channel such that the delay between the two channels is as close to 0 as possible.

See also.

Unsupported Probe dialog on page 200

Probe Compensation configuration menu (analog channels Probe Setup panel) on page 200

Other panel (Channel configuration menu) on page 201

Deskew configuration menu (Other panel, Channel configuration menu) on page 201

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. If Off, tap the **AFG** button on the Settings bar. When Output is set to On, the oscilloscope changes the AFG button to an AFG badge that shows the AFG settings.
2. If On, double-tap the AFG badge to open the AFG menu.

Arbitrary/Function Generator overview

The function generator provides output of predefined waveforms up to 50 MHz. Choose between Sine, Square, Pulse, Ramp, DC, Noise, Sin(x)/x, Gaussian, Lorentz, Exponential Rise, Exponential Decay, Haversine, Cardiac and Arbitrary signals.

You can select a predefined waveform or load a saved .wfm-format waveform from storage (USB drive or network). You can also use a .CSV (spreadsheet) waveform file stored externally.

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 Out.

Field or control	Description
Output	Tap to toggle the output On or Off.
Waveform Type	Tap to select an available waveform from the list.
Load	Tap to open the Directory configuration menu. Navigate to and select a waveform file to load into the AFG memory. Only shown when Waveform Type = 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. Only shown when the Waveform Type = Ramp.
Width	Sets the width of the pulse using the keypad or multipurpose knob. Only shown when the Waveform Type = Pulse.
Duty Cycle	Sets the duty cycle of the square wave using the keypad or multipurpose knob. Only shown when the Waveform Type = Square.
Frequency	Sets the frequency of the waveform using the keypad or multipurpose knob. The frequency range is 0.1 Hz to 50 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
Offset	Sets 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

Field or control	Description
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.

AFG output amplitude ranges by impedance (peak to peak)

Waveform	50 Ω	High Z
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	0 V to ±1.25 V	0 V to ±2.5 V
Noise	10 mV to 2.5 V	20 mV to 5 V
Sine(x)/x	10 mV to 1.5 V	20 mV to 3 V
Gaussian	10 mV to 1.25 V	20 mV to 2.5 V
Lorentz	10 mV to 1.2 V	20 mV to 2.4 V
Exponential Rise	10 mV to 1.25 V	20 mV to 2.5 V
Exponential Decay	10 mV to 1.25 V	20 mV to 2.5 V
Haversine	10 mV to 1.25 V	20 mV to 2.5 V
Cardiac	10 mV to 2.5 V	20 mV to 5 V
Arbitrary	10 mV to 2.5 V	20 mV to 5 V

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 **Add New 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	Enter label text in this field. The default 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).
Bus Type	Select a bus from the drop down list. The Parallel bus type comes standard on the instrument. Serial buses require purchase and installation of serial bus triggering and analysis options. See *Serial bus and trigger options* on page 11.
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.
Display format	Enables showing just the decoded bus or both the bus and its digital waveforms. You can also tap on the + symbol on the bus waveform to toggle between showing the bus only or showing bus and source waveforms.
Decode format	Sets how decoded data information is shown in the bus. Select from listed formats. Available formats depend on the bus type.

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

Bus type configuration menus

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

FlexRay serial bus configuration menu on page 213

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

ARINC 429 serial bus menu

Use the ARINC 429 bus menu to set up and decode a ARINC 429 avionics network serial data bus waveform. Triggering on and decoding of ARINC 429 bus type requires option 5-SAERO (installed at factory) or SUP5-SAERO (field upgrade).

To set up the ARINC 429 avionics serial data bus:

- Tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the new Bus badge. Set the **Bus Type** to **ARINC429**.
- To change the settings on an existing ARINC 429 serial bus waveform, double-tap the **Bus** waveform badge and make necessary changes.

ARINC 429 serial bus menu fields and controls.

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text using an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **ARINC429**.
Polarity	Select the polarity to match the ARINC 429 bus being acquired.
Source	Select the ARINC 429 signal source.
High Threshold, Low Threshold	Sets the valid high and low threshold values for the signal source.
Bit Rate	Sets the bit rate to 12,500, 100,000, or Custom.
Custom Rate	Sets a custom data bit rate. To set the value, tap the field and use the Multipurpose knob, double-tap the field and use the Custom Rate virtual keypad, or double-tap the field and use an attached keyboard. Only available when Bit Rate = Custom.
Data Format	Sets the data format to Data (19 bits), SDI (Source/Destination Identifiers) plus data (21 bits), or SDI plus Data plus Sign/Status Matrix (SSM) (23 bits.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and Mixed Hex.

Other bus types. Other serial bus types, such as CAN, LIN, Ethernet, and so on, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

Audio serial bus configuration menu

Use the Audio bus menu to set up and display Audio Type I2S, Left Justified (LJ), Right Justified (RJ), or TDM Audio serial bus waveforms. Triggering on and decoding of Audio bus types requires option 5-SRAUDIO (installed at factory) or SUP5-SRAUDIO (upgrade).

To set up the Audio serial bus:

- To create a new Audio bus waveform, tap the **Add New Bus** button on the Settings bar and open the Bus configuration menu. Set the **Bus Type** to **Audio**.
- To change the settings on an existing Audio serial bus waveform, double-tap the **Bus** waveform badge and make the necessary changes.

Audio serial bus menu fields and controls.

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **Audio**.
Audio Type	Sets the digital audio signal type. Select from the drop-down list.
Bit Order	Set the waveform to decode with most-significant (MS) bit first or least-significant (LS) bit first.
Bit Clock	Set the signal source, logic level threshold, and polarity for the Bit Clock signal.
Word Select	Set the signal source, logic level threshold, and normal or invert signal setting for the Word signal.
Data	Set the signal source, logic level threshold, and logic definition (active high or low) for the Data signal.
Word Size (Audio Type = I2S, LJ, or RJ)	Set the number of bits used in a Word for the selected audio type (8, 12, 16, 18, 20, 24, 28, or 32-bits).
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information.
TDM-specific settings	
Frame Sync	Set the signal source, logic level threshold, and polarity for the frame sync signal.
Data Bits per Channel	Set the number of data bits per audio channel.
Clock Bits per Channel	Set the number of clock bits per audio channel.
Channels per Frame	Set the number of audio channels per data frame.
Bit Delay	Sets the bit delay (number of bits).

Other bus types. Other serial bus types, such as CAN, USB, I2C, FlexRay, and so on, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

FlexRay serial bus configuration menu on page 213

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

CAN serial bus configuration menu

Use the CAN bus menu to set up and display a CAN (Controller Area Network) or CAN FD serial bus waveform. Triggering on and decoding of CAN bus type requires option 5-SRAUTO (installed at factory) or SUP5-SRAUTO (upgrade).

To set up the CAN serial bus:

- To create a new CAN bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **CAN**.
- To change the settings on an existing CAN serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar for the CAN bus to open the configuration menu.

CAN serial bus menu fields and controls.

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **CAN** to set up and display a CAN bus waveform.
Signal Type	Sets the CAN signal type to decode. Default is **CAN_H**.
CAN Standard	Sets the CAN signal standard to decode. Default is **CAN 2.0**.
Source	Select the signal source from listed analog and digital channels.
Threshold	Sets the high/low logic transition level.
Sample Point	Sets the sample point from 5% to 95% of the position within the bit period or the unit interval.
Bit Rate	Select the bit rate of your CAN bus serial data. To enter a custom bit rate, select Custom and enter the custom bit rate in the Custom Rate input box. Only available when **CAN Standard** = **CAN 2.0** .
SD Bit Rate	Select the SD bit rate of your CAN FD serial bus data. To enter a custom bit rate, select Custom and enter the custom bit rate in the Custom Rate input box. Only available when **CAN Standard** = **CAN FD (ISO)** or **CAN FD (non-ISO)**.
FD Bit Rate	Select the FD bit rate of your CAN FD serial bus data. To enter a custom bit rate, select Custom and enter the custom bit rate in the Custom Rate input box. Only available when **CAN Standard** = **CAN FD (ISO)** or **CAN FD (non-ISO)**.
Custom rate	Sets the custom bit rate to use to decode the signal. Tap the field and use the Multipurpose knob to change the value, or double-tap on the field and use the virtual keypad to enter a custom bit rate. Only available when **Bit Rate** or **SDI Bit Rate** = **Custom.**
Display Format	Bus sets the waveform view to show just the decoded bus information. Bus and Waveform sets the waveform view to show both the decoded bus and the source signal waveforms. You can also tap on the + symbol on the bus waveform to toggle between showing the bus only or showing bus and source waveforms.

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

Other bus types. Other serial bus types, such as RS232, LIN, I2C, FlexRay, and so on, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

Ethernet serial bus menu on page 211

FlexRay serial bus configuration menu on page 213

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

Ethernet serial bus menu

Use the Ethernet bus menu to set up and display an Ethernet 10BaseT or 100BaseT serial bus waveform. Triggering on and decoding of Ethernet serial bus type requires option 5-SRENET (installed at factory) or SUP5-SRENET (upgrade).

To use the Ethernet serial bus menu:

- To create a new Ethernet bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **Ethernet**.
- To change the settings on an existing Ethernet serial bus waveform, double-tap the Ethernet **Bus** waveform badge to open the configuration menu.

Ethernet serial bus menu fields and controls.

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **Ethernet**.
Speed	Sets the network signal speed.
Signal Type	Sets the signal type to Single Ended or Differential.
Source	Sets the signal source for Differential. Only available when Signal Type = Diff.
D+ Input, **D- Input**	Defines the signal sources and threshold values for the Single Ended data+ and - signals. Only available when Signal Type = Single Ended.
High Threshold	Sets the threshold value for a logic 1 value. Only available when Signal Type = Diff.
Low Threshold	Sets the threshold value for a logic 0 value. Only available when Signal Type = Diff.
IPv4	Set to Yes if the Ethernet signal being measured uses Internet Protocol version 4 (IPv4).
Q-(VLAN)	Set to Yes if the Ethernet signal being measured uses IEEE 802.1Q virtual LANs.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, Mixed ASCII, and Mixed Hex.

Other bus types. Other serial bus types, such as CAN, LIN, I2C, FlexRay, and so on, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

FlexRay serial bus configuration menu on page 213

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

FlexRay serial bus configuration menu

Use the Flexray bus menu to set up and display a Flexray automotive network serial bus waveform. Triggering on and decoding of Flexray bus types requires option 5-SRAUTO (installed at factory) or SUP5-SRAUTO (upgrade).

To set up the FlexRay serial bus:

- To create a new FlexRay bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **FlexRay**.
- To change the settings on an existing FlexRay serial bus waveform, double-tap the **Bus** waveform badge and make necessary changes.

FlexRay serial bus menu fields and controls.

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text using an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **FlexRay**.
Signal Type	Select the FlexRay signal type being measured.
Channel Type	Set to A or B channel.
Source	Select the FlexRay signal source.
Threshold	Sets the threshold value for the TX or RX signal type.
High Threshold, Low Threshold	Sets the high and low threshold values for the BM Inverted and Bdiff/BP signal types.
Bit Rate	Select a bit rate. To set a custom bit rate, select Custom and enter a value in the Custom Rate field.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types. Other serial bus types, such as CAN, LIN, Ethernet, and so on, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

I2C serial bus configuration menu

Use the I2C bus menu to set up and display an I^2C (Inter-Integrated Circuit) serial bus waveform. Triggering on and decoding of I^2C bus type requires option 5-SREMBD (installed at factory) or SUP5-SREMBD (upgrade).

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

- To create a new I^2C bus waveform, tap the **Add New 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	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
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.
Include R/W bit in Address	Select **Yes** to display 7-bit addresses as eight bits, where the eighth bit (LSB) is the R/W bit, or display 10-bit addresses as 11 bits, where the third bit is the R/W bit. Select **No** to display 7-bit addresses as seven bits, and 10-bit addresses as ten bits.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types. Other serial bus types, such as CAN, LIN, SPI, FlexRay, and so on, are available as purchasable options. Once purchased and installed, the new bus types are shown in the **Bus Type** menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

FlexRay serial bus configuration menu on page 213

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

LIN serial bus configuration menu

Use this menu to set up and display a LIN (Local Interconnect Network) serial bus waveform. Triggering on and decoding of LIN bus type requires option 5-SRAUTO (installed at factory) or SUP5-SRAUTO (upgrade).

To set up the LIN serial bus:

- To create a new LIN bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **LIN**.
- To change the settings on an existing LIN serial bus waveform, double-tap the LIN **Bus** waveform badge and make necessary changes.

LIN serial bus menu fields and controls.

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **LIN**.
Source	Set the signal source from available analog or digital channels.
Threshold	Set the threshold level to define a logic high level.
Polarity	Select the polarity to match the LIN bus being acquired.
LIN Standard	Select the standard to match the LIN bus being acquired.
Bit rate	Sets the bit rate. To enter a custom bit rate, tap Custom and enter the custom bit rate in the Custom Rate input box.
Include Parity Bits with ID	Set to Yes to include parity bits with the ID.
Sample Point	Sets the sample point from 5% to 95% of the position within the bit period or the unit interval.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and Mixed.

Other bus types. Other serial bus types, such as CAN, Audio, I2C, FlexRay, and so on, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

FlexRay serial bus configuration menu on page 213

I2C serial bus configuration menu on page 215

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

MIL-STD-1553 serial bus menu

Use the MIL-STD-1553 bus menu to set up and decode a MIL-STD-1553 aeronautic network serial data bus waveform. Triggering on and decoding of MIL-STD-1553 bus type requires option 5-SAERO (installed at factory) or SUP5-SAERO (field upgrade).

To set up the MIL-STD-1553 serial data bus:

- To create a new MIL-STD-1553 bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **MIL-STD-1553**.
- To change the settings on an existing FlexRay serial bus waveform, double-tap the **Bus** waveform badge and make necessary changes.

MIL-STD-1553 serial bus menu fields and controls.

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text using an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **MIL-STD-1553**.
Polarity	Select the polarity to match the MIL-STD-1553 bus being acquired.
Source	Select the MIL-STD-1553 signal source.
High Threshold, Low Threshold	Sets the valid high and low threshold values for the signal source.
RT Maximum	Sets the maximum valid response time (RT) for a command.
RT Minimum	Sets the minimum valid response time (RT) for a command.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, Mixed ASCII, and Mixed Hex.

Other bus types. Other serial bus types, such as CAN, LIN, Ethernet, and so on, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

Parallel Bus configuration menu

Use this menu to set up and display a parallel bus waveform. Parallel bus decoding and triggering is included with the oscilloscope.

To set up the parallel bus:

- To create a new parallel bus, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **Parallel**.
- To change the settings on an existing parallel bus waveform, double-tap the **Bus** waveform badge to open the configuration menu, and make necessary changes.

Parallel bus configuration menu fields and controls.

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to Parallel to define a parallel bus.
Clocked Data	Toggles Yes or No to use a clock signal to recover the data bits from the bus inputs.
Clock Source	Sets the source for the bus clock signal. The source can be an analog or digital channel. Only available when Clocked Data is set to Yes.
Polarity	Sets the clock signal edge to use for timing reference. Only available when Clocked Data is set to Yes.
Threshold	Sets the threshold value to determine high logic value. Only available when Clocked Data is set to Yes.
Define Inputs	Opens a Parallel Bus - Define Inputs configuration menu to set the signal sources and the bit order (MSB to LSB) for the bus. See *Parallel Bus - Define Inputs menu* on page 221.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types. Serial bus types, such as CAN, Ethernet, USB, and FlexRay, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

FlexRay serial bus configuration menu on page 213

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

Parallel Bus - Define Inputs menu

Use this menu to select the signal sources and order for the parallel bus waveform

To access the Parallel Bus - Define Inputs menu, double-tap a Parallel Bus badge to open the configuration menu, and tap the **Define Inputs** button.

Parallel Bus - Define Inputs menu fields and controls.

Field or control	Description
Parallel bus definition list	Lists the signal source and thresholds of selected channels or waveforms. The MSB is at the top of the list. To add a signal to the Parallel bus definition list, tap a source button in the Sources list. The button moves from the Sources list to the bottom of the bus list. Use the arrow buttons to the right of the field to move a selected signal up or down in the list. To remove a signal from the Parallel bus (and return it to the Sources list), tap on the signal source button. To change the threshold value for individual channels, tap in a selected Threshold field and use the assigned multipurpose knob, or double-tap the field to open the keypad and enter values.
Sources	Lists all available sources to use for a parallel bus. To add a source to the Parallel bus definition list, tap a source button. The button moves from the Sources list to the bottom of the bus list.
Set All Thresholds	Sets all thresholds in the Parallel bus definition list to the specified value. Enter a value and tap Apply to set the values.

RS232 serial bus menu

Use this menu to set up and display an RS232 serial bus waveform. Triggering on and decoding of RS232 buses requires option 5-SRCOMP (installed at factory) or SUP5-SRCOMP (upgrade).

To set up a RS232 serial bus:

- To create a new RS232 bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **RS232**.
- 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	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **RS232**.
Bit Rate	Sets the data bit rate. To enter a custom bit rate, tap Custom and enter the custom bit rate in the Custom Rate input box.
Source	Set the signal source from available analog or digital 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.
Data Bits	Set the number of bits that define a data packet for your RS232 bus.
Parity	Set the parity to match the RS232 bus being acquired.
Packet View	Set to On to show decoded packet level information on the bus waveform.
End of packet	select the appropriate end of packet value to match the RS232 bus being acquired. Available when Packet View = On.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and ASCII.

Other bus types. Other serial bus types, such as CAN, USB, I2C, and FlexRay, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

FlexRay serial bus configuration menu on page 213

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

SPI serial bus configuration menu on page 223

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

SPI serial bus configuration menu

Use the SPI bus menu to set up and display an SPI (Serial Peripheral Interface) synchronous serial bus waveform. Triggering on and decoding of SPI bus type requires option 5-SREMBD (installed at factory) or SUP5-SREMBD (upgrade).

To set up the SPI serial bus:

- To create a new SPI bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **SPI**.
- 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	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
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 = SS.
SDA Input	Select the channel source and threshold level for the Serial Data signal (bidirectional). Set the Polarity to use Active High or Active Low logic for the SDA signal.
Word Size	Enter the word size, in bits. Common word sizes are 8, 16, and 32.
Bit Order	Set to most significant bit first (MS) or least significant bit first (LS).
Idle Time (Framing =)	Set the idle frame time. Available when Framing = Idle.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types. Other serial bus types, such as CAN, LIN, I^2C, and FlexRay, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

FlexRay serial bus configuration menu on page 213

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

USB serial bus configuration menu on page 225

See also.

Bus Trigger configuration on page 288

USB serial bus configuration menu

Use the USB bus menu to set up and display an USB 2.0 (Universal Serial Bus) waveform. Triggering on and decoding of USB buses requires option 5-SRUSB2 (installed at factory) or SUP5-SRUSB2 (upgrade).

To set up a USB serial bus:

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

USB serial bus menu fields and controls.

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **USB**.
Speed	Set the speed to match the USB bus you are acquiring. Selecting High Speed (480 Mbps) sets the Signal Type to Differential.
Signal type	Set to match the USB signal you are acquiring (Single Ended or Differential). Use a differential probe to acquire the differential USB signal. Selecting High Speed (480 Mbps) sets the Signal Type to Differential. Both Single Ended and Diff can be used to measure Full (12 Mbps) and Low (1.2 Mbps) speed USB signals.
Source	Select the channel source for the signal from a differential probe. Only available when Signal Type = Diff.
High Threshold	Set the high threshold level for the differential signal.
Low Threshold	Set the low threshold level for the differential signal.
D+ Input	Select the channel source and set the threshold level for the Data+ signal. Only available when Signal Type = Single Ended.
D- Input	Select the channel source and set the threshold level for the Data- signal. Only available when Signal Type = Single Ended.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types. Serial bus types, such as CAN, LIN, I2C, and FlexRay, are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu. The serial bus options also add corresponding bus trigger capabilities to the **Trigger** menu.

ARINC 429 serial bus menu on page 206

Audio serial bus configuration menu on page 207

CAN serial bus configuration menu on page 209

Ethernet serial bus menu on page 211

FlexRay serial bus configuration menu on page 213

I2C serial bus configuration menu on page 215

LIN serial bus configuration menu on page 216

MIL-STD-1553 serial bus menu on page 218

Parallel Bus configuration menu on page 219

RS232 serial bus menu on page 221

SPI serial bus configuration menu on page 223

See also.

Bus Trigger configuration on page 288

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
2. Touch and hold a cursor readout or cursor line and select **Configure Cursors** from the right-click menu.

To display cursors on the screen :

1. Tap the **Add New... Cursors** button of the display, or
2. Push the **Cursors** front-panel button to toggle cursors on or off.

Cursor configuration menu fields and controls

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

Field or control	Description
Display	Tap to toggle the cursor display On or Off.
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 crosses a waveform. **V Bars** are vertical cursors that measure horizontal parameters (typically time). They are not associated with the waveform, but simply show the cursor time position 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 simply show the cursor amplitude position in the vertical scale. **V&H Bars** cursors measure vertical and horizontal parameters simultaneously. They are not associated with the waveform, but simply show the cursor time and amplitude position.
Source Waveform	Tap to select the source waveform (the waveform on which to display the cursors). **Same** places both cursors on the same waveform. **Split** allows each cursor to be on a different waveform. Only available when Cursor Type = Waveform.
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. Only available when Source Waveform = Same.
Cursor A Source, Cursor B Source	Tap to select the waveform sources for Cursor A and B. Only available when Source Waveform = Split.
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 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.
Time Zone	Tap and select the desired time zone from the drop-down list.
Automatically adjust clock for Daylight Saving Time	Sets the oscilloscope to automatically change the time to account for Daylight Saving Time changes.

NOTE. *You cannot set a specific time; the time is preset at the factory.*

Digital channel configuration menu

Use the Digital channel menu to enable individual digital channels, set their thresholds, and add labels.

To open the Digital channel configuration menu, double-tap a Digital channel badge. You can also double-tap on the digital channel handles to open the menu.

Digital channel settings fields and controls

Field or control	Description
Display	Tap to toggle display of the channel On and Off.
Height	Sets the relative height of the digital waveform on the screen. Only available when Waveform View mode = Overlay.
D7-D0 **Bit**	Tap to toggle individual channels (bits) on or off and remove them from the display.
D7-D0 **Threshold**	Sets threshold level values for individual data channels.
D7-D0 **Label**	Enter label text for individual data channels. The label is shown to the right of the corresponding digital channel.
Turn All Off	Turns off the digital channel and removes the Digital channel badge from the bar.
Set All Thresholds	Use to set all data channels to the same threshold value. Enter a value in the field and tap the button.

See also

Analog channel configuration menu

DVM configuration menu

Use this menu to set up the optional digital voltmeter (DVM) function to use probes to measure AC, DC, or AC+DC voltages.

To open the digital voltmeter menu:

1. If the DVM is Off, tap the **DVM** badge on the Settings bar. 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 DMV 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.
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. The DVM can only measure analog channels.
Mode	Tap to select measuring DC, AC RMS, or DC+AC RMS measurement modes.
Show Basic Statistics in Badge	Tap to toggle showing DVM measurement statistics in the DVM badge.

Menu bar overview

The Menu bar provides access to file, utility, and help functions.

The Menu bar

Field or control	Description
File	Provides typical system file management operations such as opening, saving, moving, and renaming files. See *Recall configuration menu (File menu)* on page 232. See *Save As configuration menu (File menu)* on page 234. See *File Utilities configuration (File menu)* on page 239. **Autoset** executes an immediate Autoset operation. See *Quickly display a waveform (Autoset)* on page 67. **Default Setup** immediately restores the oscilloscope to factory default settings. See *Using Default Setup* on page 91. **Shutdown** powers off the oscilloscope.
Edit	Provides a menu to Undo or Redo the last operation. See *Undo, Redo (Edit menu)* on page 241.
Utility	Use to set user preferences, configure input, output, and network settings, run self tests, verify calibration status and run signal path compensation, and erase nonvolatile memory. See *User Preferences (Utility menu)* on page 241. See *I/O (Utility menu)* on page 243. See *Self Test configuration menu (Utility menu)* on page 247. See *Calibration configuration menu (Utility menu)* on page 248. See *Security configuration menu for standard instruments (Utility menu)* on page 248.
Application	Use to access installed software applications. This menu only appears when testing applications are installed on the oscilloscope. Only valid for 5-WIN or SUP5-WIN installations. See *Application (Menu bar)* on page 241.
Help	Opens the Help viewer, provides a tutorial to teach touch screen fundamentals, and displays current instrument software and option license information. See *Help... (Help menu)* on page 253. See *User Interface Tutorial (Help menu)* on page 253. See *About (Help menu)* on page 253.

Recall configuration menu (File menu)

Use this menu to recall (load) reference waveforms, instrument setups, and sessions (setup plus waveforms).

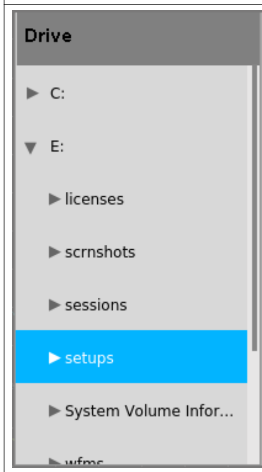
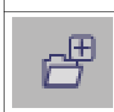
To open the file Recall configuration menu:

1. Tap **File** on the menu bar.
2. Tap **Recall** to open the Recall configuration menu.

File operations and Option 5-WIN (Microsoft Windows 10 Operating System SSD) 5 Series MSO only (not MSO58LP). Instruments with Option 5-WIN installed (Windows 10 SSD) will display the standard Windows file tools to navigate to and select files and folders.

The Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. This is different from standard instruments (without 5-WIN installed), which assign a non-changing drive letter to each USB port.

Recall configuration menu fields and controls.

Field or control	Description
Look in:	Shows the current directory path to the location of a file. 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 virtual 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, up to a maximum of 20 locations.
File type to open (tabs)	Tabs on the left let you set which type of file to recall (Waveform, Setup, or Session). Selecting a file type sets the file extensions in the Files of Type field to the correct value.
	The Drive column lists the directory structure, opening at the root (/) level. Use to quickly navigate to a location. Tap to list the contents of the directory in the Name pane. Double-tap an item to display the directory and any subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
	Use the arrow buttons to navigate the file directory. 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.
	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.
Files of type	Use to select the file format you want to open. The drop-down list shows all file extension types that the instrument can read for the selected file type.
Cancel	Cancels the file open action and closes the configuration menu.
Recall	Recalls the selected file. Recalling a waveform file adds a Reference waveform badge to the Settings bar and displays the waveform as it fits in the current Horizontal setting. Recalling a Setup or Session immediately sets the oscilloscope to the settings in the file and displays any waveforms that were part of the recalled session.

USB port drive names and locations.

Save As configuration menu (File menu)

Use this menu to configure saving screen captures, waveforms, oscilloscope setups, reports, and session files.

To access the Save As configuration menu, tap **File** on the menu bar and select **Save As...**.

NOTE. *Selecting File > Save the first time after powering up the oscilloscope opens the Save As configuration menu. This lets you set or verify the save locations for all types of information you can save.*

Once the Save As configuration menu has been opened and closed, the next time you select Save causes the instrument to automatically save the file type last selected in the Save As configuration menu. This lets you quickly save files with few steps.

Saving files with the front-panel User button. Pushing the front-panel **User** button automatically saves the file type last selected in the Save As configuration menu. If no saves have been performed since the instrument power-up, pushing the User button opens the Save As configuration menu. Select the type of save operation you want to perform and tap OK. After that, pushing the User button automatically saves the file type.

NOTE. *The User button is not assigned a specific save type; it saves whatever save action was last selected in the Save As configuration menu.*

File operations and Option 5-WIN (Microsoft Windows 10 Operating System SSD) 5 Series MSO only (not MSO58LP). Instruments with Option 5-WIN installed (Windows 10 SSD) will display the standard Windows file tools to navigate to and select files and folders.

The Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. This is different from standard instruments (without 5-WIN installed), which assign a non-changing drive letter to each USB port.

Save As configuration menu fields and controls. The following fields and controls are common to all Save As actions.

Field or control	Description
File save type	Tabs on the left let you set which type of file to save (Screen Capture, Waveform, Setup, Report, or Session). Selecting a file type sets the file extensions in the Save As Type field to the correct value.
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 file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual 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.
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* on page 237 .
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. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. 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.
Auto Increment File Name	Select to enable auto-incrementing of the file name. Auto increment adds a number count to the end of the file name, and increments it on each subsequent saving of the same file name.
Count	Sets the increment count start number. Default is 000.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the 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 configuration menu, and displays a confirmation message.

Screen Capture tab fields and controls. The following settings are specific for saving a screen capture

Field or control	Description
File save type	Tap the Screen Capture tab to save a screen image to a file. Selecting Screen Capture sets the file extensions in the Save As Type field to available graphic file formats.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the graphic save format.

Waveform tab fields and controls. The following settings are specific for saving a waveform.

Field or control	Description
File save type	Tap the Waveform tab to save waveform(s) to a file. Selecting Waveform sets the file extensions in the Save As Type field to available waveform file formats.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the graphic save format.
Source	Sets the waveform source to save. You can save a single waveform, or save all active (displayed) waveforms. Waveform file names include the source channel or waveform appended to the end of the file name.

Setup tab fields and controls. The following settings are specific for saving an instrument setup.

Field or control	Description
File save type	Tap the Setup tab to save the instrument setup and measurement settings to a file. Selecting Setup sets the file extension in the Save As Type field to .set.
Include Reference Waveforms	Include the instrument waveforms.

Report tab fields and controls. The following settings are specific for saving an instrument report.

A report can include information on instrument settings, measurement results, screen images, individual measurement configuration, source input settings, and error and warning information for measurements. Reports are either a PDF file or a single file Web page.

Field or control	Description
File save type	Tap the Report tab to save a report file. Selecting Report sets the file extensions in the Save as Type field to available report file formats.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the report save format (PDF or Single File Web Pages).
Append Report	Appends a report to an existing report file.
Comments	Add comments to clarify the contents or purpose of the report, or specifics of the signals being measured.
Include Images and Annotations	Add Waveform and Plot images to the report.
Include Setup Configuration	Add instrument and measurement configuration information to the report.
Include Error/Warnings Log	Add any error or warning logs to the report.

Session tab fields and controls. The following settings are specific for saving an instrument session.

A session is a zipped file that contains an instrument setup file and all acquired waveform data. Use session file contents to move analysis activities to a PC and free up the instrument for others to use.

Field or control	Description
File save type	Tap the Session tab to enable saving a session file. Selecting Session sets the file extensions in the Save As Type field to .tss.
Save As Type	A session file can only be saved to format .tss.

Browse Save As Location configuration menu

Use this menu to select a new file location to save files.

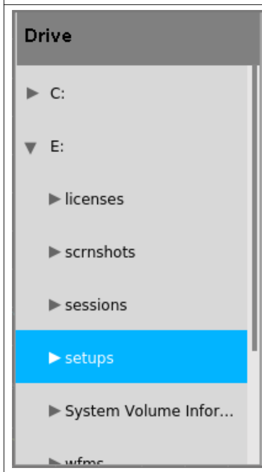
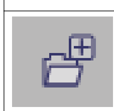
To access the Browse Save As Location configuration menu:

1. Select **File > Save As...**.
2. In the Save As configuration menu, tap the **Browse** button.

File operations and Option 5-WIN (Microsoft Windows 10 Operating System SSD) 5 Series MSO only (not MSO58LP). Instruments with Option 5-WIN installed (Windows 10 SSD) will display the standard Windows file tools to navigate to and select files and folders.

The Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. This is different from standard instruments (without 5-WIN installed), which assign a non-changing drive letter to each USB port.

Browse Save As Location configuration menu fields and controls.

Field or control	Description
File path field	Shows the current directory. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recently accessed file save paths, for the current save type, up to a maximum of 20 paths.
	The Drive column lists the directory structure, opening at the root level. Use to quickly navigate to a location. Tap to list the contents of the directory in the Name pane. 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. Drag the list up and down to show more entries.
	Use the arrow buttons to navigate the file directory. 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.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
Cancel	Cancels the file path change, closes the configuration menu, and returns to the Save As configuration menu. No file path change occurs.
OK	Closes the configuration menu, returns to the Save As configuration menu, and updates the Save Location field with the path from the Browse configuration menu.

USB port drive names and locations.

File Utilities configuration (File menu)

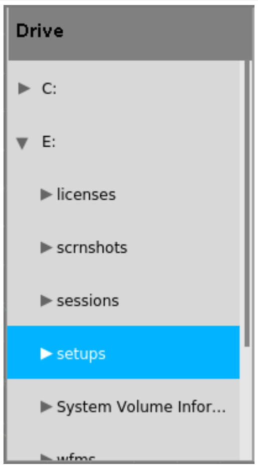
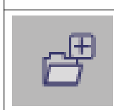
Use this menu to copy, paste, delete, and rename files, and unmount memory devices from USB ports.

To access the File Utilities configuration menu, select **File > File Utilities...** from the Menu bar.

File operations and Option 5-WIN (Microsoft Windows 10 Operating System SSD) 5 Series MSO only (not MSO58LP).
Instruments with Option 5-WIN installed (Windows 10 SSD) will display the standard Windows file tools to navigate to and select files and folders.

The Windows operating system assigns the first available drive letter (typically E:) to the first USB device attached to the oscilloscope, regardless of which port the USB device is plugged into. The next plugged-in USB device is assigned the next available drive letter (such as F:) and so on for other installed devices. This is different from standard instruments (without 5-WIN installed), which assign a non-changing drive letter to each USB port.

File Utilities configuration menu fields and controls.

Field or control	Description
File path field	Shows the current directory. 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 virtual 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.
	The Drive column lists the directory structure, opening at the root level. Use to quickly navigate to a location. Tap to list the contents of the directory in the Name pane. Double-tap an item to display the directory and any subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
	Use the arrow buttons to navigate the file directory. 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.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
Copy	Copies the selected file in the filename pane to memory.
Paste	Pastes the file from the last Copy action in the current File Utilities session into the current location.
Delete	Deletes the selected file or folder.
Rename	Renames the selected file or folder.
Unmount	Closes the file writing session on the attached device to let you disconnect the device from the USB port. Select the drive port letter and tap Unmount. You can now remove the USB device from the port.

USB port drive names and locations.

Undo, Redo (Edit menu)

The Edit menu lets you Undo or Redo recent actions. Tap Undo or Redo. Not all oscilloscope actions can be undone. If the Undo or Redo menu items are grayed out, then the last action cannot be undone or redone.

Application (Menu bar)

Use this menu to access installed software applications. This menu only appears when testing applications are installed on an oscilloscope running the Microsoft Windows operating system (option 5-WIN or SUP5-WIN).

The Application menu bar item is located between the Edit and Utility menus. Use the drop-down list to select the application software to run. See the application software help for information on using the application.

User Preferences (Utility menu)

Use this menu to set global display, measurement, jitter and eye analysis settings, and other user preferences.

To open the User Preferences menu:

1. Tap the **Utility** menu.
2. Tap **User Preferences** to open the configuration menu.
3. Tap a panel.

Display panel fields and controls.

Field or control	Description
Colors	Tap and select either Normal or Inverted colors. Inverted makes the waveform background white, with graticule marking in black. Inverted does not change the Menu bar, Results bar, or Settings bar colors.
Default Waveform View Display Mode	Tap and select the default waveform view display mode, either Overlay or Stacked.
Backlight	Tap to select the intensity of the backlight. ***NOTE.*** *Performing an Autoset resets the backlight value to High.*
Auto-Dim	Select On to automatically dim the screen backlight after a specified time.
Time	Sets the amount of time to wait before the display is dimmed. Tap in the field and use the knob to change the time value, or double-tap to open the virtual keypad and set a time value. Only available when Auto-Dim is On.

Measurements panel fields and controls.

Field or control	Description
Shared Reference Levels	Tap to select either Global or Per Source reference levels. **Global** applies the same reference levels to all measurement sources. **Per Source** allows selection of a different reference levels for all measurement sources.
Calculate Reference Levels	Tap to select how often to calculate reference levels. **First Acq** only calculates the reference levels on the first acquisition and re-uses them for all subsequent acquisitions. Pushing the Clear front-panel button, or making measurement changes (adding, deleting, or reconfiguring) also clears the reference levels which are then recalculated on the next acquisition. **Every Acq** calculates the reference levels with every acquisition.
Measurement Interpolation	Interpolation defines how the waveform is drawn between waveform record data points. Tap and select the measurement interpolation method. **Auto** selects the best interpolation method. **Sin(x)/x** interpolation computes record points using a curve fit between the actual samples acquired. **Linear** interpolation computes record points between actual acquired samples by using a straight line fit.
Measurement Annotations	Annotations show the exact segment of the waveform from which the measurement is derived. The annotation types consist of horizontal bars, vertical bars, or cross-hatch marks. **Auto** sets annotations to display if valid for the measurement. To view annotations for a measurement, select that measurement badge. If annotations are valid for that measurement, they are added to the waveform source for that measurement. **Off** turns off display of measurement annotations.

Jitter and Eye Analysis panel fields and controls.

Field or control	Description
Calculate Reference Levels	Tap to select calculating the reference levels on the first acquisition or on every acquisition.
Horizontal Measurement Units	Tap to select horizontal measurement units of seconds or unit intervals.
Jitter Separation Model	Tap to select the jitter separation model (Spectral Only Spectral + BUJ).
Lock RJ Value	Selecting the Lock RJ Value calculates the measurements at the specified random jitter value. The checkbox is unchecked by default. Selecting the checkbox displays a text box where you can enter the RJ value. The default value is 1ps. Lock RJ Value cannot be configured when Jitter Separation Model is Spectral + BUJ.

Other panel fields and controls.

Field or control	
Right Clicks via Touch	Turns On or Off the ability to use touch and hold method to open right-click menus on badges and other screen items.
Time	Sets the time it takes to respond to a touch and hold before opening a right-click menu.

I/O (Utility menu)

Use this configuration menu to set up a LAN, USB Device Port, Socket Server, and AUX OUT signal parameters.

To open the I/O menu:

1. Tap the **Utility** menu.
2. Tap **I/O...**.

Entering and applying LAN network changes.

When first opening the I/O menu LAN panel, the Network Address is set to **Auto** (default setting) and the **Apply Changes** button is grayed out (inactive) in the LAN panel.

When you select any editable input box and start entering data, the **Apply Changes** button becomes active, and the characters being entered are bolded and italicized. Bold italicized text means that the values have not been applied to the oscilloscope settings.

When you tap the **Apply Changes** button, all changes are saved (takes about 10 seconds), the text is changed to normal font (nonbold, nonitalic), and the **Apply Changes** button becomes inactive.

If you tap outside the I/O menu before you tap the Apply Changes button, the menu closes and none of your changes are saved.

LAN panel fields and controls.

Field or control	Description
LAN Status	A readout that indicates the status of the LAN connection, either a Green circle with the word Normal or a Red circle with an error message.
Host Name	The instrument host name is displayed. To change the name, double-tap and enter a name in the virtual keyboard.
Network Address	Tap to select Manual or Automatic mode. In Automatic mode, the current Instrument IP Address, Gateway IP Address, Subnet Mask, and DNS IP Address are displayed.
Domain Name	The instrument domain name is displayed. To change the name, double-tap and enter a name in the virtual keyboard.
Instrument IP Address	Tap and use the multipurpose knobs to enter the address. Use the A knob to select the digit, and the B knob to change the value. Only available to edit when Network Address = Manual
Subnet Mask	Tap and use the multipurpose knobs to enter the mask. Use the A knob to select the digit, and the B knob to change the value. Only available to edit when Network Address = Manual
Service Name	The instrument service name is displayed. To change the name, double-tap and enter a name in the virtual keyboard.
Gateway IP Address	Tap and use the multipurpose knobs to enter the address. Use the A knob to select the digit, and the B knob to change the value. Only available to edit when Network Address = Manual
DNS IP Address	Tap and use the multipurpose knobs to enter the address. Use the A knob to select the digit, and the B knob to change the value. Only available to edit when Network Address = Manual
MAC Address	A readout of the instrument MAC Address. This field is not editable.
e*Scope HTTP Port	A readout of the instrument e*Scope HTTP port number. This field is not editable.
Test Connection	Tap to test the connection. If the connection test is successful, then **OK** is displayed. If the test is unsuccessful, then **No Response** is displayed.
LAN Reset	Tap to display the *LAN Reset configuration menu (Utility > I/O menu)* on page 246.
Apply Changes	Apply changes made on this panel to the instrument. ***NOTE.*** *No changes are made to instrument settings until you tap the Apply Changes button.*

USB Device Port fields and controls. Use the USB Device Port panel to enable or disable USB ports and set the GPIB Talk/ Listen address. Use USB ports to connect a USB memory device, keyboard, or for direct PC control of the oscilloscope using USBTMC protocol.

Field or control	Description
USB Device Port	Tap to toggle all USB device ports On and Off.
USBTMC Configuration	Displays the USBTMC configuration information.
GPIB Talk/Listen Address	Double-tap and enter the address using the virtual keypad.

Socket Server panel fields or controls. Use the following socket server settings to set up and use a socket server between your oscilloscope and a remote terminal or computer.

Field or control	Description
Socket Server	Tap to toggle the socket server On or Off.
Protocol	Tap to select a protocol, either None or Terminal. A communication session run by a user at a keyboard typically uses a terminal protocol. An automated session might handle its own communications without such protocol from the oscilloscope.
Port	Enter the port number using the multipurpose knob or virtual keypad.

AUX Out panel fields and controls. Use the following settings to select the signal that is output on the rear-panel AUX Out signal connector.

Field or control	Description
AUX Out Signal	Tap to select the signal type to send to the AUX Out connector. **Trigger** sends a pulse for each trigger occurrence. **Reference Clock** outputs the instrument's reference signal, whether generated internally or supplied from the REF In connector. **AFG Sync** sends a pulse that is synchronized to the AFG output signal.
Polarity	Tap to select the polarity of the trigger signal (positive or negative pulse per trigger event). The polarity control is only present for the Trigger output.

Using Telnet to communicate with the oscilloscope .

1. After you have set up the socket server parameters, the computer is now ready to communicate with the oscilloscope. If you are running an MS Windows PC, you could run its default client Telnet, which has a command interface. One way to do this is by typing `Telnet` in the Run window. The Telnet window will open on the PC.

   ***NOTE.** On MS Windows 10, you must first install Telnet.*

2. Start a terminal session between your computer and your oscilloscope by typing in an open command with the oscilloscope's LAN address and port number.

   You can obtain the LAN address by pushing the Ethernet & LXI lower menu button and the resulting LAN Settings side menu button to view the resulting Ethernet and LXI Settings screen. You can obtain the port # by pushing Socket Server on the lower menu and viewing Current Port on the side menu.

   For example, if the oscilloscope IP address was 123.45.67.89 and the port # was the default of 4000, you could open a session by writing into the MS Windows Telnet screen: o 123.45.67.89 4000.

   The oscilloscope will send a help screen to the computer when it has finished connecting.

3. You can now type in a standard query, such as, *idn?.

   The Telnet session window will respond by displaying a character string describing your instrument.

   You can type in more queries and view more results using this Telnet session window. You can find the syntax for relevant commands, queries and related status codes in the Programmer Manual that is available at the Tektronix website.

   ***NOTE.** Do not use the computer's backspace key during an MS Windows Telnet session with the oscilloscope.*

LAN Reset configuration menu (Utility > I/O menu)

Use this menu to reset the Local Area Network (LAN) settings to the listed default settings.

To open the LAN Reset dialog:

1. Tap **Utility** in the Menu bar.
2. Tap **I/O...** .
3. Tap the **LAN Reset** button to open the LAN Reset configuration menu.
4. Tap **OK** to reset the LAN settings.
5. Tap **Cancel** to close the dialog without taking any action, and return to the I/O configuration menu.

LAN Reset default settings.

Function	Setting
Network address	Automatic
DHCP	Enabled
BOOTP	Enabled
mDNS & DNS-SD	Enabled
e* Scope Password Protection	Disabled
LXI Password Protection	Disabled
e* Scope and LXI Password	Empty string (default)

See also.

I/O (Utility menu) on page 243

Self Test configuration menu (Utility menu)

Use this menu to view power-on diagnostic results, run self tests, and verify 250K termination control on input channels.

To open the Self Test configuration menu:

1. Tap **Utility** in the Menu bar
2. Tap **Self Test...** .

To close the Self Test menu, tap anywhere outside of the menu.

NOTE. *Remove all input signals before running the self test.*

Self Test configuration menu fields and controls.

Field or control	Description
Results	The status of the overall test and each individual test is displayed.
Test Mode	Tap and select the test loop mode from the drop-down: **Run N Loops** runs all self tests the specified number of times, then stops. **Run Until Failing** continuously runs tests until a failure occurs, then stops. **Run Always** continuously runs tests without stopping for failures. You must tap the Abort Self Test button to stop testing. **Loop on Failure** continuously runs tests until a failure occurs, then continuously runs the failed self test. The status of the failed test stays at Running. You must tap the Abort Self Test button to stop testing. Changing the Test Mode does not stop any ongoing self tests. You must abort a test run before the new test mode will be run.
N =	Enter the number of times to run (loop) the Run N Loops test.
Run Self Test / Abort Self Test	Tap to run all standard power-on self tests. While tests are running, the button changes to Abort Self Test. When self tests are stopped, the button reverts to Run Self Test. Tap the Abort Self Test button anytime to stop testing.

250K Ohm Termination Verification dialog. Tap a button to verify that 250K Ohm termination can be enabled or disabled for a channel. Closing the dialog restores normal termination setting of applicable probes.

Calibration configuration menu (Utility menu)

Use this menu to perform a signal path compensation or view the factory calibration status.

To open the Calibration configuration menu:

1. Tap **Utility** in the Menu bar
2. Tap **Calibration...** .

Calibration configuration menu fields and controls.

Field or control	Description
Factory Adjustment Status	This area at the top of the menu lists the instrument calibration status. Factory Adjustment Status should be Calibrated. If an instrument becomes uncalibrated, this displays an Uncalibrated status and a red Warning message bar is shown at the top of the screen in the Menu bar area. Contact your nearest Tektronix Service Center for assistance.
SPC Status	Indicates the status of the last SPC run (Pass or Failed). Also indicates how long ago the last SPC was run.
Run SPC	Signal path Compensation (SPC) corrects for internal DC inaccuracies caused by temperature variations and/or long-term drift in circuits. ***NOTE.*** *SPC takes 5 or more minutes per channel to run.* Allow the instrument to warm up for 20 minutes before running SPC. Remove all probes, cables, and adapters from the input connectors before running SPC. Tap Run SPC to run signal path compensation. The compensation can take up to five minutes per channel.

Security configuration menu for standard instruments (Utility menu)

If you have acquired confidential data, use TekSecure® to erase the oscilloscope memory before you return the oscilloscope to general use. This menu applies to all standard instruments that were not configured with Option 5-SEC (Enhanced Security) when ordered.

NOTE. *This menu applies to all standard instruments that were not ordered with Option 5-SEC (Enhanced Security) when purchased. For instruments with Option 5-SEC installed, see* *Security configuration menu (Option 5-SEC instruments only) on page 249.*

To run the Security process:

NOTE. *Save any important waveform, screen capture, instrument setup, report, and session files to external memory before running TekSecure. All such files will be erased.*

1. Tap **Utility** in the Menu bar.
2. Tap **Security...** .
3. Tap **Run TekSecure** to erase nonvolatile memory. It will take approximately seven minutes to erase the memory.
4. Push the **Default Setup** front panel button to load memory with the instrument factory settings.

NOTE. *You cannot stop the TekSecure process once it is started.*

NOTE. *TekSecure does not erase calibration constants or instrument firmware.*

TekSecure Erase Memory

Use this menu to use the TekSecure® function to erase the oscilloscope nonvolatile memory.

To open the Security menu and run TekSecure:

1. Tap **Utility** in the Menu bar.
2. Tap **Security...** .
3. Tap **Run TekSecure** to run the TekSecure memory erase. It will take approximately seven minutes to erase the memory.
4. To exit the dialog without running TekSecure, tap outside of the configuration menu.

NOTE. *You cannot stop the TekSecure process once it is started.*

NOTE. *Do not power down the instrument while TekSecure is running.*

NOTE. *Save any important waveform, screen capture, instrument setup, report, and session files to external memory before running TekSecure.*

NOTE. *TekSecure does not erase calibration constants or instrument firmware.*

Security configuration menu (Option 5-SEC instruments only)

Use this menu to enable or disable USB ports, LAN port, Firmware updates, and securely erase the oscilloscope nonvolatile memory (TekSecure®). Access to the functions on this menu are password protected. This menu is only shown on instruments with Option 5-SEC installed.

To open the Security configuration menu:

1. Tap **Utility** in the Menu bar
2. Tap **Security**.
3. Tap **On** or **Off** to enable or disable a port or network. The default setting in On.
4. Tap **Set Password** to enter a password for the first time. If a password has been set, tap **Change Password** to change the password.

NOTE. *The switches cannot be toggled (off or on) unless a password has been set.*

- *If a switch is toggled without a password set, a message is shown to set a password.*
- *If a switch is toggled with a password set, the instrument shows the Enter Password menu.*

Security configuration menu fields and controls.

Field or control	Description
USB Ports	Tap to enable (On) or disable (Off) all USB ports (Device and Host).
LAN Port	Tap to enable (On) or disable (Off) the Ethernet port.
Firmware Updates	Tap to enable (On) or disable (Off) the ability to update the oscilloscope firmware.
Set Password	Opens the *Set Password* menu.
Change Password	Opens the *Change Password* menu.
TekSecure Erase Memory	Tap Run TekSecure to erase nonvolatile memory. It will take approximately seven minutes to erase the memory. ***NOTE.*** *You cannot stop the TekSecure process once it is started.* ***NOTE.*** *Do not power down the instrument while TekSecure is running.* ***NOTE.*** *Save any important waveform, screen capture, instrument setup, report, and session files to external memory before running TekSecure.* ***NOTE.*** *TekSecure does not erase calibration constants or instrument firmware.*

Passwords and sending the instrument for service. If you need to send the instrument for service or repair, make sure to use the **Clear Password** function in the Change Password menu before sending to the Tektronix Service Center. Failure to do so may impede servicing the instrument.

Enter Password configuration menu (Option 5-SEC instruments only)

Use this function to enter the password to access the Option 5-SEC security functions. This menu is only shown on instruments with Option 5-SEC installed.

To enter the password to allow changing the selected security feature state (On or Off):

1. Tap **Utility** in the Menu bar.
2. Tap **Security**.
3. Tap the button of a function that you want to change, to open the Enter Password menu.
4. Enter the password and tap **OK**. The menu closes and toggles the selected function's button state.

Enter Password configuration menu fields and controls.

Field or control	Description
Password	Enter the password. ***NOTE.*** *If the instrument has a keyboard attached, and you have disabled the USB ports, double-tap on the password field to open the virtual keyboard and enter the password.*
OK	Closes the menu and toggles the selected function's button when the entered password is correct. Any other conditions result in an error message.
Cancel	Closes the menu without taking any action.

Passwords and sending the instrument for service. If you need to send the instrument for service or repair, make sure to use the **Delete Password** function in the Change Password menu before sending to the Tektronix Service Center. Failure to do so may impede servicing the instrument. See *Change Password configuration menu (Option 5-SEC instruments only)* on page 252.

Set Password configuration menu (Option 5-SEC instruments only)

Use this function to set the password used to access the Option 5-SEC security functions. This menu is only shown on instruments with Option 5-SEC installed.

To access the Set Password configuration menu:

1. Tap **Utility** in the Menu bar.
2. Tap **Security**.
3. Tap **Set Password**.

Set Password menu fields and controls.

Field or control	Description
New Password	Enter the new password in this field. [1]
Repeat New Password	Re-enter the new password in this field.[3]
Set New Password	Sets the new password and closes the menu when the new password matches in both new password fields and the new password fields are not empty. Any other conditions result in an error message.
Cancel	Closes the menu without taking any action.

Passwords and sending the instrument for service. If you need to send the instrument for service or repair, make sure to use the *Clear Password* function in the Change Password menu before sending to the Tektronix Service Center. Failure to do so may impede servicing the instrument.

[1] If the instrument has a keyboard attached, and you have disabled the USB ports, double-tap on the password field to open the virtual keyboard and enter the password.

Change Password configuration menu (Option 5-SEC instruments only)

Use this function to change an existing password used to access the Option 5-SEC security functions. This menu is only shown on instruments with Option 5-SEC installed.

To access the Change Password configuration menu:

1. Tap **Utility** in the Menu bar
2. Tap **Security**.
3. Tap **Change Password**.

Change Password menu fields and controls.

Field or control	Description
Current Password	Enter the current password in this field. [2]
New Password	Enter the new password in this field.[4]
Repeat New Password	Re-enter the new password in this field.[4]
Delete Password	Deletes the password from the security function. You must enter the correct password before you can clear it. If the password is correct, the password is deleted and the menu closes. The main Security menu changes the Change Password button to **Set Password**. If the incorrect password is entered, the instrument shows an error message. ***NOTE.*** *Once you delete the password, you must set a new password before you can change the security access functions.*
Set New Password	Sets the new password and closes the menu if the current password is correct, the new password matches in both new password fields, and the new password fields are not empty. Any other conditions result in an error message.
Cancel	Closes the menu without taking any action.

Passwords and sending the instrument for service. If you need to send the instrument for service or repair, make sure to use the Clear Password function before sending to the Tektronix Service Center. Failure to do so may impede servicing the instrument.

Demo (Utility menu)

Use this menu to access demonstrations of key oscilloscope features.

To open the Demo configuration menu, select **Utility > Demo...** in the Menu bar.

[2] If the instrument has a keyboard attached, and you have disabled the USB ports, double-tap on the password field to open the virtual keyboard and enter the password.

Demo menu fields and controls.

Field or control	Description
Demo overview pane	The upper half of the menu shows an overview of the demonstration available in the selected panel. This pane may also contain a screen shot showing the waveforms and capability being demonstrated.
Connection Details	Tap this button to open a dialog box that shows how to connect the oscilloscope to a signal source to perform the selected demonstration.
Recall Demo Session	Tap this button to load the session file for the selected demonstration.
Demo panels	Each panel provides a demonstration of oscilloscope capabilities within a category. The categories are Miscellaneous and Serial Bus. Each panel will have two or more demonstration buttons in them. Selecting a button updates the upper half of the menu to show the relevant content (and image if available) for the selected demonstration.

Help... (Help menu)

Tap this item to open the 5 Series MSO Help viewer. This Help viewer is similar in operation to a traditional Microsoft Windows help viewer.

User Interface Tutorial (Help menu)

Tap this menu item to run a screen tutorial to learn the basics of the touch screen user interface.

To start the tutorial animation, tap **Help > User Interface Tutorial**. While the tutorial is running, you do not have access to any scope actions.

The tutorial closes automatically when it completes. You can also close the tutorial anytime by tapping on the Close button in the text box.

About (Help menu)

Use the About configuration menu to show instrument information and installed options, and to install or uninstall analysis or feature options.

To open the About menu:

1. Tap **Help** on the menu bar.
2. Select **About** from the menu to open the About configuration menu.

About configuration menu fields and controls.

Field or control	Description
System information	Provides system-related information such as model, bandwidth, serial number, Host ID, and installed firmware version. Provide this information when communicating with Tektronix to purchase option licenses or communicate with Customer Support.
Probes Detected	Lists probes connected to the instrument. TekVPI probes will list the probe model, serial number, and installed probe firmware version. Non-TekVPI probes may show their attenuation factor. ***NOTE.*** *Connecting or disconnecting probes while the About menu is open does not update the Probes Detected list. The Probes Detected list is not dynamic.*
Installed Options	Lists installed options. Click on an item in the list to show details of the option in the Option Details area below the list.
Option Details	Lists details of the option selected in the list, including license type, when the license was checked out, when it was installed on the instrument, and the date the license expires.
Remove License	Opens the Location to Save the Exit Key dialog. Use this to uninstall a floating license option for use by others. See *Location to Save Exit Key configuration menu* on page 254.
Install License	Tap this button to open the Browse License Files dialog to navigate to and select an option license file to install. See *Browse License Files menu (Help > About)* on page 256.

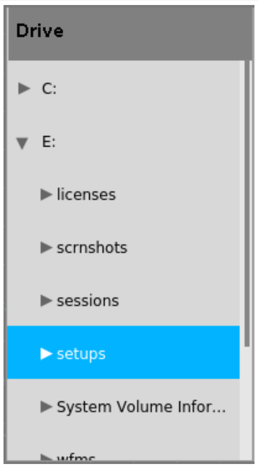
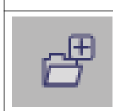
Location to Save Exit Key configuration menu

Use this menu to navigate to and set the location to save the option license key when you uninstall the license.

To access the Location to Save Exit Key menu:

1. Tap **Help** on the menu bar.
2. Select **About** from the menu.
3. Tap a license in the list that you want to uninstall (return the license).
4. Tap the **Remove License** button to open the Location to Save Exit Key configuration menu.

Location to Save Exit Key configuration menu fields and controls.

Field or control	Description
Look in	Shows the current directory path at which to save the exit key file. Tap on the file path and use a keyboard to enter a new path. 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 recently accessed paths, up to a maximum of 20.
	The Drive column lists the directory structure, opening at the root level. Installed USB memory devices are listed. Use to quickly navigate to a location. Tap to list the contents of the directory in the Files pane. 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. Drag the list up and down to show more entries. You can also use the files Name column to navigate to and select a folder in which to save the file.
	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.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File Name	Lists the selected license file name to return (uninstall). Tap the down arrow icon on the right end of the field to open a list of uninstalled license 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 Name field (where folder and files are listed) only lists files of the specified type.
Cancel	Cancels the license uninstall process, closes the configuration menu without saving any changes that were made, and returns to the About configuration menu.
Create	Closes the configuration menu and saves the license information to the specified location.

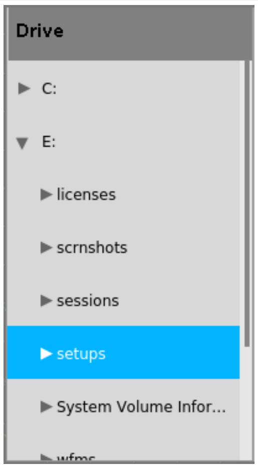
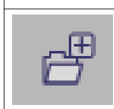
Browse License Files menu (Help > About)

Use this menu to select and install an option license file to enable new functions.

To access the Browse License Files menu:

1. Tap **Help > About...** on the menu bar.
2. Tap **Install License** button to open the Browse License Files configuration menu.

Browse License Files configuration menu fields and controls.

Field or control	Description
Look in	Shows the current directory path and file name. Tap on the file path and use a keyboard to enter a new path. 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 recently accessed files, up to a maximum of 20.
	The Drive column lists the directory structure, opening at the root level. Installed USB memory devices are listed. Use to quickly navigate to a location. Tap to list the contents of the directory in the files listing. 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. 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.
	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 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 the new 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 pane only lists files of the specified type.
Cancel	Cancels the configuration menu changes, closes the menu, and returns to the prior menu without making any changes.
Open	Closes the configuration menu, returns to the About configuration menu, and installs the license. Follow any instructions that may be shown during the installation.

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 Mode	Tap to select either Automatic or Manual horizontal mode. In **Automatic** mode you can set the **Minimum Sample Rate** and **Horizontal Scale**. In **Manual** mode you can set the Sample Rate and Record Length. Horizontal Scale is a dependent variable calculated from the sample rate and record length.
FastAcq	Tap to toggle **FastAcq** mode On or Off. **FastAcq** provides high-speed waveform capture. It is helpful in finding elusive signal anomalies. Fast Acq mode reduces the dead time between waveform acquisitions, enabling the capture and display of transient events such as glitches and runt pulses. Fast Acq mode can also display waveform phenomena at an intensity that reflects their rate of occurrence.
FastAcq Palette	Tap and select the desired FastAcq Palette from the drop-down list. **Temperature** uses color-grading to indicate frequency of occurrence, with hot colors like red/ yellow indicating frequently occurring waveform data points, and colder colors like blue/green indicating rarely occurring data points. **Spectral** uses color-grading to indicate frequency of occurrence with, colder colors like blue indicating frequently occurring waveform data points, and hot colors like red indicating rarely occurring data points. **Normal** uses the default channel color (like yellow for channel one) along with gray-scale to indicate frequency of occurrence where frequently occurring events are bright. **Inverted** uses the default channel color (like yellow for channel one) along with gray-scale, but inverts color intensity such that events that occur less frequently are brighter. Only available when FastAcq is On.
Minimum Sample Rate	Sets the minimum sample rate for acquisitions. Changing horizontal scale will not reduce the sample rate below this value. This can result in partial waveform records at lower frequencies, but at the same time would provide more sample points for a few cycles of the signal. Available when Horizontal Mode = Automatic and Fast Acq = Off. This setting can be overridden if **Allow Horizontal Scale to Override Min Sample Rate** is selected.
Allow Horizontal Scale to Override Min Sample Rate	Select to set the oscilloscope to automatically change the sample rate to acquire a full waveform record. Changes to the sample rate are shown in the Horizontal badge readout, not in the Minimum Sample rate field of the menu. Only available when Horizontal Mode = Automatic and Fast Acq = Off.

Field or control	Description
Maximum Record Length	Tap and select the Maximum Record Length from the drop-down list. Record length is dependent on the Scale setting. If changing the scale causes the record length to exceed the maximum record length Limit, the sample rate is decreased to the next available setting to keep the record length below the set maximum. Only available when Horizontal Mode = Automatic and FastAcq = On.
Allow Horizontal Scale to Override Max Record Length	Select to set the oscilloscope to automatically change the record length to acquire a full waveform record. Changes to the sample rate are shown in the Horizontal badge readout, not in the Minimum Sample rate field of the menu. Only available when Horizontal Mode = Automatic and FastAcq = On.
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.
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 (pretrigger) or after the trigger point (posttrigger).
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 pretrigger and posttrigger 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 s	Tap to set the delay position to 0 s (center of the waveform record. Only available when Delay = On.
Set to 10%	Tap to set the trigger delay to 10% of the waveform record. Only available when Delay = Off.
Sample Rate Changes Affect	Tap to allow changes to the Sample Rate to affect either the Horizontal Scale or the Record Length. Only available when Horizontal Mode = Manual.

Field or control	Description
Sample Rate	Tap to set the Sample Rate using the assigned multipurpose knob, double-tap to set the rate using the virtual keypad, or tap the up and down arrows. This keeps the oscilloscope at the specified sample rate regardless of the horizontal or record length settings. This limits the available horizontal scale settings to those that can be used with the specified sample rate. Only available when Horizontal Mode = Manual.
Record Length	Tap to set the Record Length using the assigned multipurpose knob, double-tap to set the length using the virtual keypad, or tap the up and down arrows. Only available when Horizontal Mode = Manual.

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 **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 on page 260

Equation Editor (Math configuration menu) on page 262

Add Filter menu (math Equation Editor) on page 263

Add Variable menu (math Equation Editor) on page 264

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 **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. Only available when Auto Scale is disabled (unchecked) and Math Type = 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 = Basic or Advanced.
Label	Enter a label for the math waveform.
Position	Sets the vertical position of the math waveform. Available when Math Type = Basic or Advanced.
Set to 0	Sets the vertical position of the math waveform to zero (vertical center of a slice (Stacked mode) or the screen (Overlay mode). Available when Math Type = Basic or Advanced.
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 = On.
Math Type	Sets the type of math waveform to display. **Basic** creates a math waveform by adding, subtracting, multiplying, or dividing two analog 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. See *Math FFT plot configuration menu (Math waveform)* on page 270. **Advanced** displays a drop-down from which to select the 20 last-accessed equations created by the Equation Editor. This mode also provides access to the Equation Editor.
Source, Source1, Source 2	Defines the signal source or sources for a Basic or FFT math waveform. Basic and FFT math waveforms are created from analog channels only (Ch, Math, or Ref). Available when Math Type = Basic or FFT.
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 = Basic.

Field or control	Description
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 20) 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. You can also double-tap on an equation in this field and directly edit the equation using the virtual keyboard. See *Equation Editor (Math configuration menu)* on page 262. Available when Math Type = Advanced
Edit	Opens the Equation Editor to create advanced math waveforms from analog channels, reference, math waveform, measurement, filter, and variable sources. Tap the Edit button to open the Equation Editor. See *Equation Editor (Math configuration menu)* on page 262. Available when Math Type = Advanced

Math waveform guidelines.

- Digital channels are not valid in math waveforms.
- 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 can 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, variables, filters, and functions.

To access the math Equation Editor:

1. Double-tap a **Math** waveform badge. If no Math badge is present, tap the **Add New Math** button 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. Drag the sources field up or down to scroll through selections. If a source column is empty, there are no active or defined sources for that category.
Add Filter	Tap to open the Add Filter menu, to create a filter definition to add to the Filters source column in the Equation Editor menu. See *Add Filter menu (math Equation Editor)* on page 263.
Add Variable	Tap to open the Add Variable menu, to add a defined variable to the Variables source column in the Equation Editor menu. See *Add Variable menu (math Equation Editor)* on page 264.
Functions	Select the math functions to apply to your signal or signals. See *Add Functions (math Equation Editor)* on page 265.
Keypad	Use to enter numeric, basic math operations, and logic conditions.
Clear	Clears the math equation field.
Arrows	Use to move the cursor position in the Math equation field.
Delete	Deletes the selected part of the equation (recommend using a mouse to most easily select equation text).
Backspace	Deletes the character to the left of the cursor.
Apply	Tap to apply the math expression to create the waveform. The math expression will populate to the Math x = input box in the Math configuration menu and also appears in the Math badge on the Settings bar.
Cancel	Closes the editor menu, does not update the equation list if you made any changes from the last time you Apply'd 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 Filter menu (math Equation Editor)

Use the Add Filter menu to add a high pass, low pass, or arbitrary filter expression to the Filters column of the Equation Editor Sources table.

To access the Add Filter menu:

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

Add Filter menu fields and controls.

Field or control	Description
Filter Type	Sets the filter type: **High Pass**, **Low Pass**, or **ArbFlt** (arbitrary filter). ArbFlt requires you to load a FLR-format filter file. If loading a filter file, tap Load to navigate to and select the FLR file to load.
Cutoff Frequency	Sets the filter cutoff frequency as a predefined fraction of the sample rate (SR). Default is 0.25 * SR.
Label	Shows the filter selections as A:B, where A is the filter type (HP or LP) and B is cutoff frequency setting. Arbitrary filter files use the file name as the label.

Add Variable menu (math Equation Editor)

Use the Add Variable menu to add a defined variable source to the Equation Editor Sources table, which you can then add to your math waveform expression.

To open the Add Variable menu:

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

Add Variable menu fields and controls.

Field or control	Description
Variable Type	Selects a predefined variable to add to the Variables column of the Equation Editor. **Sample Rate** creates a variable with the value of the current sample rate. **Sample Interval** creates a variable with the value of the current sample interval (1/sample rate) **Record Length** creates a variable with the value of the current record length.
Cancel	Closes the menu without adding a variable to the **Equation Editor** Variables column.
OK	Closes the menu and adds the current variable selection to the Variables column of the **Equation Editor's Sources** table.

Add Functions (math Equation Editor)

Use the Add Functions controls to add predefined math operations to you 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.
Max	Inserts the text MAX(into the math expression. The MAX function accumulates over time the maximum value at each point in the vector.
Log	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.
e^y	Natural antilog. Inserts the text EXP(into the math expression. The EXP function produces the natural antilog of the argument.
⌈x⌉	Smallest integer. Inserts the text CEIL(into the math expression. The CEIL function takes the largest integer > the expression.
Invert	Inserts the text INV(into the math expression. Enter an argument or channel to the function. The invert function inverts the argument within the parentheses.
FFT Mag	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.
FFT Phs	FFT Phase. Inserts the text FftPhase(into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT math waveform that shows the phase components of the source signal.
d/dx	Derivative. Inserts the text DIFF(into the math expression. Enter an argument to the function. The derivative function produces the derivative of the argument.
Min	Inserts the text MIN(into the math expression. The MIN function accumulates over time the minimum value at each point in the vector.
LogE	Natural logarithm. Inserts the text LN(into the math expression. The natural logarithm function produces the natural logarithm of the argument.
√x	Inserts the text SQRT(into the math expression. Enter an argument to the function.
⌊x⌋	Largest integer. Inserts the text FLOOR(into the math expression. The FLOOR function takes the largest integer < the expression.

Button	Description
|x|	Absolute. Inserts the text FABS(into the math expression. The FABS function takes the absolute value of the expression.
FFT Real	FFT Real. Inserts the text FftReal(into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT math waveform that displays only the real part of the source signal.
FFT Img	FFT Imaginary. Inserts the text FftImaginary(into the math expression. Select one of the waveforms as an argument to the function. This function creates an FFT math waveform that displays only the imaginary part of the source signal.
SIN	Inserts the text SIN(into the math expression.
COS	Inserts the text COS(into the math expression.
TAN	Inserts the text TAN(into the math expression.
SIN^{-1}	Arc sine. Inserts the text ASIN(into the math expression.
COS^{-1}	Arc cosine. Inserts the text ACOS(into the math expression.
TAN^{-1}	Arc tangent. Inserts the text ATAN(into the math expression.
Deg	Degrees. Inserts the text DEG(into the math expression. The function expresses the value of the expression in degrees.
Rad	Radians. Inserts the text RAD(into the math expression. The function expresses the value of the expression in Radians.

Font Color menu (Text Settings configuration)

Use this menu to change the label color. Touch and hold on note or label text, select **Format Text** in the right-click menu, and select **Color** to open this menu. Click on a color to change the text color.

Text Settings configuration menu (Note and Waveform labels text)

Use this menu to change and format existing Note or Waveform labels (font type and size, color, bold, italic, and underline).

Prerequisite: There is note or waveform label on the screen.

To open the **Text Settings** configuration menu, double-tap on the Note or Waveform label. You can also touch and hold on the note/label text and select **Format Text** from the right-click menu.

To delete a Note or label, touch and hold on the note/label and select **Delete** from the right-click menu.

Text Settings configuration menu fields and controls

Field or control	Description
Text	Double-tap and enter the desired text using the virtual keyboard.
Font	Tap and select the desired font from the drop-down list.
Size	Tap and select the desired font size from the drop-down list.
Color	Tap and select the desired font color from the color palette.
B	Tap to toggle text bolding On or Off.
I	Tap to toggle text italics On or Off.
U	Tap to toggle text underlining On or Off.

Plot configuration menus

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

Use the following links to access information on a specific Plot configuration menu.

Eye Diagram plot configuration menu on page 268

Math FFT plot configuration menu (Math waveform) on page 270

Histogram plot configuration menu on page 273

Bathtub plot configuration menu on page 274

Spectrum plot configuration menu on page 275

Trend Plot configuration menu on page 277

Plot XY configuration menu on page 278

XYZ plot configuration menu on page 278

Harmonics Bar Graph plot configuration menu (5-PWR, SUP5-PWR, or 5-PS2 only) on page 279

SOA plot configuration menu (5-PWR, SUP5-PWR, or 5-PS2 only) on page 280

Trajectory plot configuration menu (Switching Loss power measurement) (5-PWR, SUP5-PWR, or 5-PS2 only) on page 281

Eye Diagram plot configuration menu

Use this menu to change settings of a displayed Eye Diagram plot.

To open the Eye Diagram plot menu, double-tap anywhere in the **Eye Diagram** plot view.

Settings panel (Eye Diagram 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 to view an area of interest. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and position.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. Only available when AutoScale is off (unchecked).
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. Only available when AutoScale is off (unchecked).
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.

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 OK 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 OK 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.

FastFrame Timestamp Trend plot configuration menu

Use this menu to change settings of a displayed FastFrame Timestamp plot.

To open the FastFrame Timestamp plot configuration menu, double-tap anywhere in a **FastFrame Timestamp** Plot view.

FastFrame Timestamp plot configuration menu fields and controls.

Field or control	Description
AutoScale	Toggles AutoScale On or Off (Default is On). Turn AutoScale off to set the X and Y-axis range to view an area of interest. When AutoScale is Off (unchecked), the Plot view zoom mode enables and a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. You can also use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Selects which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is Off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is Off.

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 OK 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 OK 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.

Math FFT plot configuration menu (Math waveform)

Use the Math FFT plot menu to change settings of a displayed FFT plot, including source, FFT window type, plot type, and gating.

To open the FFT plot configuration menu, double-tap anywhere in the **Math FFT** Plot view.

Plot Settings panel (Math FFT plot configuration menu) fields and controls.

Field or control	Description
Auto Scale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot (Horizontal, Vertical, or Both).
X-Axis Scale	Sets the horizontal frequency scale to either Log or Linear. A **Log** scale is useful when the frequency component magnitudes cover a wide dynamic range, letting you show both lesser and greater- magnitude frequency components on the same display. A **Linear** scale is useful when the frequency component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Y-Axis Scale	Sets the vertical amplitude scale to either Log (dBm) or Linear. A **Log dB** scale is useful when the frequency component magnitudes cover a wide dynamic range, letting you show both lesser and greater magnitude frequency components on the same display. A **Linear** scale is useful when the frequency component magnitudes are all close in value, enabling direct comparison of their magnitudes.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scale. These values also define the horizontal zoom area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scale. These values also define the vertical zoom area shown in the small Zoom window. Available when AutoScale is off.

FFT Settings panel (Math FFT plot configuration menu) fields and controls.

Field or control	Description
FFT Type	**Magnitude** plots the magnitude values of the frequency components. **Phase** plots the phase of the signal as a function of frequency.
Window	Sets the FFT window type to use for the waveform plot. See *FFT windows* on page 272.
Gating	Sets the region of the waveform to analyze for the FFT plot. **None** uses the entire waveform record to create the FFT plot. **Screen** uses the part of the waveform record displayed on the screen (such as in Zoom mode) to create the FFT plot. **Cursors** uses the waveform data between the cursors to create the FFT plot.
Vertical Units	Sets the vertical scale to Degrees, Radians, or Group Delay. ***NOTE.*** *The Vertical Units set when FFT Type = Phase override the Y-Axis scale setting in the Plot Settings panel.* Only available when FTT Type set to Phase.
Phase Wrap	When checked, the phase trace is unwrapped where the trace jumps more than the number of degrees set in the adjacent field. Only available when FTT Type set to Phase.
Squelch	When checked, the phase trace excludes points that have a voltage smaller than the specified voltage. The squelch voltage should be set to the expected noise voltage level. Minimum value is 100 mV, and increments in units of 100 mV. When unchecked, the phase trace includes all values. Only available when FTT Type set to Phase.

Save panel (Math FFT plot configuration menu) fields and controls.

Field or control	Description
Save Plot Image	Opens the *Save As* configuration menu to specify the location and name at which to save an image file of the Plot view.
Save Plot Data	Opens the *Save As* configuration menu to specify the location and name at which to save the Plot view data to a CSV file.

FFT windows. Each FFT window is a trade-off between frequency resolution and magnitude accuracy. What you want to measure and your source signal characteristics help determine which window to use. Use the following guidelines to select the best window.

FFT window	Characteristics	Best for measuring
Hanning	Better frequency, poorer magnitude accuracy than Rectangular. Hanning has slightly poorer frequency resolution than Hamming.	Sine, periodic, and narrow-band random noise. Transients or bursts where the signal levels before and after the event are significantly different.
Rectangular	Best frequency, worst magnitude resolution. This is essentially the same as no window.	Transients or bursts where the signal levels before and after the event are nearly equal. Equal-amplitude sine waves with frequencies that are very close.
Hamming	Better frequency, poorer magnitude accuracy than Rectangular. Hamming has slightly better frequency resolution than Hanning.	Sine, periodic, and narrow-band random noise. Transients or bursts where the signal levels before and after the event are significantly different.
Blackman-Harris	Best magnitude, worst frequency resolution.	Predominantly single frequency signals to look for higher order harmonics.
Kaiser-Bessel	Less spectral leakage than the Hanning, Hamming, or Rectangular windows.	Predominantly single frequency signals to look for higher order harmonics.
Gaussian	The time-domain shape of an exponential Gaussian function transforms into a Gaussian exponential shape in the frequency domain.	Optimal localization in both the time and frequency domain.
Flattop2	Wider resolution bandwidth but lower side lobe attenuation. Also, it is unique because the time domain shape has negative values.	Useful for high accuracy magnitude measurements for signals that do not require very narrow bandwidth
TekExp	The Tek Exponential window was invented at Tektronix. In the time domain, it is not asymmetrical bell shape as is the case with the other windows. Instead, it is exponential with a peak at the 20% position of the time domain gate. The frequency domain shape is triangular. More of the acquired data record length is used to capture the impulse response.	Use this window for impulse-response testing where the 20% position is the zero phase reference point.

You can also determine the best window empirically by first selecting the Rectangular window, and then selecting (in the following order) the Hamming, Hanning, and Blackman-Harris windows until the frequency components merge. Use the window just prior to where the frequencies emerge for the best compromise between resolution and amplitude accuracy.

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.

Histogram plot configuration menu

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

To open the Histogram plot configuration menu, double-tap anywhere in a **Histogram** Plot view.

Settings panel (Histogram plot configuration menu) fields and controls.

Field or control	Description
AutoScale	Toggles AutoScale on or off. Turn AutoScale off to set the X and Y-axis range to view an area of interest. When AutoScale is off, the Plot view zoom mode enables and a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Selects which gridlines to show in the plot.
X-Axis Number of Bins	Sets the resolution by the number of bins into which the X axis is divided.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis Scale	Sets the Y axis scale to Linear or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

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 OK 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 OK 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.

Bathtub plot configuration menu

Use this menu to change settings of a displayed jitter versus BER (bathtub) plot.

To open the Bathtub plot configuration menu, double-tap anywhere in the **Bathtub** plot view.

To close a plot view, tap the X in the upper right corner of the view. Deleting the Measurement badge that opened the plot also closes the plot.

Settings panel (Bathtub plot configuration menu) fields and controls.

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Units	Sets the X-axis units to be unit intervals or seconds.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in BER units. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis scale	Sets the Y axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

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 OK 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 OK 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.

Spectrum plot configuration menu

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

To open the Spectrum plot configuration menu, double-tap anywhere in the **Spectrum** plot view.

Settings panel (Spectrum plot configuration menu) fields and controls.

Field or control	Description
AutoScale	Toggles Autoscale on or off. Turn AutoScale off to set the X and Y-axis range to view an area of interest. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Use the grid style best suited to your measurement needs.
X-Axis Scale	Sets the X axis scale to Linear or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Y-Axis Scale	Sets the Y axis scale to Linear or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Dynamic Range	Sets the vertical scale dynamic range. Available when Y-Axis Scale is set to Log.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.

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 OK 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 OK 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.

Trend Plot configuration menu

Use the Trend configuration menu to set trend waveform vertical scale and position, as well as add a label.

To open the Trend configuration menu, double-tap the **Trend** badge in the Settings bar. You can also double-tap the Trend Plot handle.

Trend configuration menu fields and controls.

Field or control	Description
Display	Toggles displaying the Trend plot on or off.
Vertical Scale	Shows the vertical scale setting (when Auto Scale is on), or sets the vertical scale (when Auto Scale is off). Use the up and own arrow buttons to set the vertical scale value when Auto Scale is turned off. You can also use the multipurpose knob or screen keypad to change the value.
Auto Scale	Enables or disables the AutoScale mode. Auto Scale uses the signal data to dynamically set the vertical scale units.
Label	Adds a label to the Trend waveform (appears next to the Trend plot handle).
Position	Shows the vertical position setting (when Auto Scale is on), or sets the vertical position (when Auto Scale is off).
Set to 0	Tap the Set to 0 button to set the trend plot to the center of the graticule.

Deleting a trend plot. To delete a Trend plot, touch and hold the **Trend** badge to open the right-click menu, and select **Delete Trend**. Deleting the Measurement badge that enabled the plot also closes the plot.

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.

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 OK 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 OK 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.

XYZ plot configuration menu

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

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

Settings panel (Plot XYZ 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.
Z-Axis	Sets the source for the Z-axis signal.

- XYZ format compares the voltage levels of the X and Y channel waveform records point-by-point, as in XY format. The displayed waveform intensity is modulated by the Z channel waveform amplitude.
- XYZ format is a dot-only display, although it can have persistence. The Vector style selection has no effect when you select XYZ format.

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 OK 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 OK 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.

Harmonics Bar Graph plot configuration menu (5-PWR, SUP5-PWR, or 5-PS2 only)

Use this menu to change settings of a displayed harmonics bar plot (Power Harmonics measurement).

To open the Bar Graph plot configuration menu, double-tap anywhere in a **harmonics** Plot view.

Settings panel (Harmonics Bar Graph 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 to view an area of interest. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Unit panel	Set the unit to Linear or Log. A Log scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A Linear scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Harmonics (Inside the Unit panel)	Select All/Odd/Even.

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 OK 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 OK 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.

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

SOA plot configuration menu (5-PWR, SUP5-PWR, or 5-PS2 only)

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

To open the SOA (XY) plot menu, double-tap anywhere in the **SOA** Plot view to open the configuration.

Settings panel (SOA 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 to view an area of interest. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
X-Axis (Voltage) Scale	To change the scale from linear to log or vice versa, click on the button (linear or log). This scales applies on the plot data as well as on the mask data.
Y-Axis (Current) Scale	To change the scale from linear to log or vice versa, click on the button (linear or log). This scales applies on the plot data as well as on the mask data.

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 OK 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 OK to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed values.

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 opened the plot also closes the plot.

Trajectory plot configuration menu (Switching Loss power measurement) (5-PWR, SUP5-PWR, or 5-PS2 only)

Use this menu to change settings of a displayed SWL (Switching Loss)Trajectory plot.

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

Settings panel (Trajectory 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 to view an area of interest. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue vertical bar in the Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and position.
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Display	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
X-Axis (Voltage) **From,To**	Sets the beginning and end scale range to display in the plot for the X-Axis scales. Only available when AutoScale is off (unchecked).
Y-Axis (Voltage) **From,To**	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. Only available when AutoScale is off (unchecked).

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 OK 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 OK to save the plot data as a comma-separated values (.csv) file. It saves the selected standard limits and computed values.

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 opened 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 file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual 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.
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. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. 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	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 file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual 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.
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. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. 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.
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.

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. Tap and enter text using a keyboard, or double-tap to open the virtual keyboard. The label text is the same color as the waveform. Once you have entered the label, close the menu and double-tap the label text to open the Text Settings menu to change the font color, size, and other characteristics.
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).

Field or control	Description
Reference File	Shows the path and file name of the current Reference waveform. Double-tap the field to open the on-screen keyboard to enter or edit the path to open a different waveform file. Tap the down arrow icon to list the 20 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)* on page 285.
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.
Selected Frame	Sets the frame to view of a recalled FastFrame waveform file. This control is only available if the recalled waveform includes FastFrame acquisitions. ***NOTE.** This control has nothing to do with the selected frame of current acquired data. Adjusting this field does not change the selected frame in acquisition data. Conversely, adjusting the selected frame in the acquisition data does not change the selected frame of the reference waveform.* Navigating through the frames of a reference waveform does not require the FastFrame feature to be active. If more than one fast frame reference waveform is active, the Selected Frame in each waveform's configuration menu is independent of the others.

Recall configuration menu (Ref waveform configuration menu)

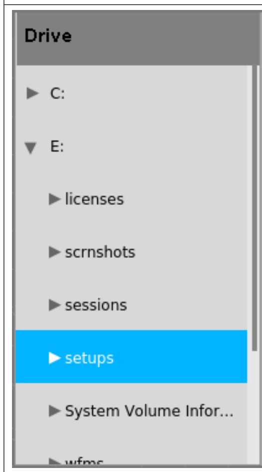
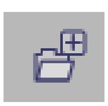
Use this menu to locate and load a reference waveform file.

Prerequisite: a Ref badge must be present on the Settings bar. See *Add a math, reference, or bus waveform* on page 77.

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. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the virtual keyboard and enter a path. 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. 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.
	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 on the file path and use a keyboard to enter a new path. 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 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.
OK	Closes the dialog, returns to the prior menu, and loads and displays the specified file.

Search configuration menu

Use the Search configuration menu to define conditions that you want to search for on a channel or waveform signal. Each occurrence of the search condition is marked on the signal with a triangle along the top of the waveform slice or view.

To open the Search menu, double-tap on a **Search** badge in the Results bar.

See the following links for information on the search type menus.

Bus Search configuration menus on page 148

Edge Search configuration menu on page 164

Logic search configuration menu on page 165

Pulse Width Search configuration menu on page 167

Rise/Fall Time Search configuration menu on page 169

Runt Search configuration menu on page 171

Setup and Hold Search configuration menu on page 172

Timeout Search configuration menu on page 173

Window Search configuration menu on page 175

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 access the Trigger menu, double-tap the Trigger badge on the Settings bar. The Trigger menu opens to show the current trigger settings.

Use the following links to see more information on specific trigger types.

- *Bus Trigger Menu*
- *Edge Trigger menu*
- *Logic Trigger*
- *Pulse Width Trigger menu*
- *Rise/Fall Time Trigger menu*
- *Runt Trigger menu*
- *Sequence Trigger menu*
- *Setup & Hold Trigger menu*
- *Timeout Trigger menu*
- *Window 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).

To open the Bus trigger menu:

NOTE. *You must add a bus to the Waveform view before you can trigger on it.* *Add a math, reference, or bus waveform on page 77.*

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.

Use the following links to access bus trigger settings information.

Bus Trigger Settings panel- ARINC 429 on page 289

Bus Trigger Settings panel - Audio bus on page 290

Bus Trigger Settings panel - CAN bus on page 291

Bus Trigger Settings panel - Ethernet bus on page 293

Bus Trigger Settings panel - FlexRay bus on page 295

Bus Trigger Settings panel - I2C bus on page 297

Bus Trigger Settings panel - LIN bus on page 298

Bus Trigger Settings panel- MIL-STD-1553 bus on page 299

Bus Trigger Settings panel - Parallel bus on page 300

Bus Trigger Settings panel - RS232 bus on page 301

Bus Trigger Settings panel - SPI bus on page 301

Bus Trigger Settings panel - USB bus on page 302

Bus Trigger Settings panel- ARINC 429. Serial bus ARINC 429 triggering is available with Option 5-SAERO.

Field or control	Description
Source	Select the **ARINC429** bus on which you want to trigger.
Trigger On	Select the type of information on which to trigger.
Trigger When Label	Trigger when the label data meets the specified condition Available when Trigger On = **Label**.
Label	Sets the label pattern on which to trigger. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Trigger On = **Label** or **Label & Data**.
Label Low	Sets the low value of the label pattern range on which to trigger. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Trigger On = **Label** and Trigger When Label = **Inside Range** or **Outside Range**.
Label High	Sets the high value of the label pattern range on which to trigger. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Trigger On = **Label** and Trigger When Label = **Inside Range** or **Outside Range**.
Data	Sets to trigger when the specified data bits condition occurs. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Trigger On = **Data** or **Label & Data**.
Data Low	Sets the low value of the data pattern range on which to trigger. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Trigger On = **Data** and Trigger When Data = **Inside Range** or **Outside Range**.
Data High	Sets the high value of the data pattern range on which to trigger. Tap the **Binary**, **Hex**, or **Octal** field and use the A and B knobs to select and change the values. Available when Trigger On = **Data** and Trigger When Data = **Inside Range** or **Outside Range**.
SSM	Sets to trigger when the specified Sign/Status Matrix (SSM) bit condition occurs. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Trigger On = **Data**.

Field or control	Description
SDI	Sets to trigger when the specified Source/Destination Identifier (SDI) bit condition occurs. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Trigger On = **Data**.
Error Type	Sets the error condition on which to trigger. Available when Trigger On = **Error**.
A, B knob controls	Use the A knob to select (highlight) the digit(s) to change in data or bit fields. Use the B knob to change the value of the digit(s) in the selected field.
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel - Audio bus.

Field or control	Description
Source	Select the **Audio** bus on which to trigger.
Trigger On	Select the type of information on which to trigger.
Data	Sets the data pattern on which to trigger. Use in conjunction with the **Trigger When** field to specify the exact trigger condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = Data.
Word	Sets the audio word channel on which to trigger (Either, Left, Right). Available when Trigger On = Data.
Trigger When	Sets the trigger when condition for the specified data pattern. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified trigger type. Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the 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).

Bus Trigger Settings panel - CAN bus.

Field or control	Description
Source	Select the **CAN** bus on which you want to trigger.
Trigger On	Select the type of information on which to trigger.
Frame Type	Sets the frame type on which to trigger. Available when Trigger On = **Type of Frame**.
Direction	Sets the transfer direction on which to trigger (Read, Write, Either). Available when Trigger On = **Identifier**.
Identifier Format	Sets the identifier for standard (11-bit) or extended (29-bit for CAN 2.0B) length. Available when Trigger On = **Identifier** or **ID & Data**.
Identifier	Sets the identifier pattern on which to trigger. The number of bits shown depends on the **Identifier Format** setting. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Trigger On = **Identifier** or **ID & Data**.
Data Bytes	Sets the number of data bytes on which to trigger (one to eight bytes). Use the A knob to change the value. Available when Trigger On = **Data** or **ID & Data**.
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. Available when Trigger On = **Data** or **ID & Data**.
Data Offset	Sets the byte offset value. Available when Trigger On = **Data** or **ID & Data**.
Trigger When	Sets the trigger when condition. Available when Trigger On = **Data** or **ID & Data**.
Bit Rate Switch (BRS) Bit	Sets the BRS bit state on which to trigger. Available when Source is a CAN FD bus and Trigger On = **FD Bits**.
Error Status Indicator (ESI) Bit	Sets the ESI bit state on which to trigger. Available when Source is a CAN FD bus and Trigger On = **FD Bits**.
Error Type	Sets the error type on which to trigger. Available when Trigger On = **Error**.
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).

Field or control	Description
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel - Ethernet bus.

Field or control	Description
Source	Select the **Ethernet** bus on which to trigger.
Trigger On	Select the type of information on which to trigger.
MAC Address Destination, Source	Sets the MAC destination and/or source address pattern on which to trigger. Tap a **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Trigger On = MAC Addresses.
MAC Length/Type	Sets the MAC length/type pattern on which to trigger. Tap a **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Trigger On = MAC Length / Type.
MAC Length/Type Low, MAC Length/Type High	Sets the boundary MAC length type conditions when testing for in-range or out of range conditions. Available when Trigger On = MAC Length / Type and Trigger When = Inside Range or Outside Range.
IP Protocol	Sets the IP protocol pattern on which to trigger. Tap a **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Trigger On = IP Header.
Source, Destination Address	Sets the source and/or destination IP address pattern on which to trigger. Tap a **Hex** or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = IP Header.
Source Port, Destination Port	Sets the source and/or destination TCP header port pattern on which to trigger. Tap a **Binary**, **Hex** or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = TCP Header.
Sequence Number	Sets the TCP header sequence number pattern on which to trigger. Tap a **Binary**, **Hex** or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = TCP Header.
Ack Number	Sets the TCP header ack number pattern on which to trigger. Tap a **Binary**, **Hex** or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = TCP Header.

Field or control	Description
Data Bytes	Sets the number of client data bytes on which to trigger (one to sixteen bytes). Use the A knob to change the value. Available when Trigger On = Client Data.
Byte Offset	Sets the client data byte offset (Don't Care or the number of bytes). Tap the field and use the A knob to change the value. Available when Trigger On = Client Data.
Client Data	Sets the data pattern on which to trigger. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the Trigger When field to specify the exact trigger condition. Tap the **Binary**, **Hex** or **ASCII** field and use the A and B knobs to select and change the values. Available when Trigger On = Client Data.
Client Data Low, Client Data High	Sets the boundary data values when testing for in-range or out of range conditions. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the Trigger When field to specify the exact trigger condition. Tap the **Binary**, **Hex** or **ASCII** field and use the A and B knobs to select and change the values. Available when Trigger On = Client Data and Trigger When = Inside Range or Outside Range.
Trigger When	Sets the trigger when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified trigger type. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Available when Trigger On = MAC Length/Type or Client 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).
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel - FlexRay bus.

Field or control	Description
Source	Select the **FlexRay** bus on which to trigger.
Trigger On	Select the type of information on which to trigger.
Indicator Bits	Select the indicator bits type on which to trigger. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Available when Trigger On = Indicator Bits or Header Fields.
Identifier	Sets the frame identifier pattern on which to trigger. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = Frame ID, Header Fields, or Identifier & Data.
Cycle Count	Sets the cycle count pattern on which to trigger. Use in conjunction with the Trigger When field to specify the exact trigger condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = Cycle Count or Header Fields.
Payload Length	Tap the Binary, Hex, or Decimal field and use the A and B knobs to select and change the values. Available when Trigger On = Header Fields.
Header CRC	Tap the Binary or Hex field and use the A and B knobs to select and change the values. Available when Trigger On = Header Fields.
Data Bytes	Sets the number of data bytes on which to trigger (one to sixteen bytes). Use the A knob to change the value. Available when Trigger On = Data or Identifier & Data.
Data	Sets the data pattern on which to trigger. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the **Trigger When** field to specify the exact trigger condition. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the values. Available when Trigger On = Data or Identifier & Data.
Byte Offset	Sets the byte offset (**Don't Care** or the number of bytes). Tap the input box and use the A knob to change the value. Available when Trigger On = Data or Identifier & Data.

Field or control	Description
Trigger When	Sets the trigger when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified trigger type. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Available when Trigger On = Frame ID, Cycle Count, Data, or Identifier & Data.
Frame Type	Sets the end of frame type on which to trigger (Static, Dynamic (DTS), ALL). Available when Trigger On = End of Frame.
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).
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel - I^2C bus.

Field or control	Description
Source	Select the **I^2C** 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, Either). Available when Trigger On = Address.
Addressing Mode	Sets the slave device address length (7 bits or 10 bits long). Available when Trigger On = **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 and B knobs to select and change the values. Available when Trigger On = **Address** or **Address & Data**.
Data Bytes	Sets the number of data bytes on which to trigger (one to five bytes). Use the A knob to change the value. Available when Trigger On = **Data** or **Address & Data**.
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. Available when Trigger On = **Data** or **Address & 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).
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel - LIN bus.

Field or control	Description
Source	Select the **LIN** bus on which to trigger.
Trigger On	Select the type of information on which to trigger.
Identifier	Sets the identifier pattern on which to trigger. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = Identifier or Identifier & Data.
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. Available when Trigger On = Data or Identifier & Data.
Trigger When	Sets the trigger when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified trigger type. Available when Trigger On = Data or Identifier & Data.
Data Bytes	Sets the number of data bytes on which to trigger (one to eight bytes). Use the A knob to change the value. Available when Trigger On = Data or Identifier & 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).
Error Type	Sets the LIN error type on which to trigger. Available when Trigger On = Error.
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel- MIL-STD-1553 bus. Serial bus MIL-STD-1553 triggering is available with Option 5-SAERO

Field or control	Description
Source	Select the **MIL-STD-1553** bus on which you want to trigger.
Trigger On	Select the type of information on which to trigger.
Sync	Sets to trigger on a Sync condition. Available when Trigger On = **Sync**.
Transmit/Receive Bit	Sets the transmit or receive bit on which to trigger. Available when Trigger On = **Command**.
Trigger When RT Address	Sets to trigger when the specified RT address condition occurs. Available when Trigger On = **Command** or **Status**.
Parity	Sets to trigger on the selected parity bit logic state. Available when Trigger On = **Command**, **Status**, or **Data**.
Address	Sets the address pattern on which to trigger. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = **Command** or **Status**.
Low Address	Sets the low value of the address pattern range on which to trigger. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger When RT Address = **Inside Range** or **Outside Range**.
High Address	Sets the high address of the address pattern range on which to trigger. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger When RT Address = **Inside Range** or **Outside Range**.
Subaddress/Mode	Sets the subaddress or mode pattern on which to trigger. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = **Command**.
Status Word Bits	Sets the status word pattern on which to trigger. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Selecting a bit shows a short description of that bit's function. Available when Trigger On = **Status**.

Field or control	Description
Data	Sets the data pattern on which to trigger. Available when Trigger On = **Data**.
Trigger When	Sets to trigger when the specified RT/IMG signal time condition occurs. Available when Trigger On = **Time (RT/IMG)**.
Minimum Time	Sets the minimum time for a valid RT/IMG signal. Available when Trigger On = **Time (RT/IMG)**.
Maximum Time	Sets the maximum time for a valid RT/IMG signal. Available when Trigger On = **Time (RT/IMG)**.
Error Type	Sets the error condition on which to trigger. Available when Trigger On = **Error**.
A, B knob controls	Use the A knob to select (highlight) the digit(s) to change in data or bit fields. Use the B knob to change the value of the digit(s).
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel - Parallel bus.

Field or control	Description
Source	Select the type of information on which to trigger.
Data	Sets the data pattern on which to trigger. The number of bits shown depends on how the parallel bus is defined. Tap the Binary or Hex field and use the A and B knobs to select and change the values.
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).
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel - RS232 bus.

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 Bytes	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 pattern on which to trigger. The number of bits shown depends on the **Data Words** setting. Tap the Binary, Hex, or ASCII field and use the A and B knobs to select and change the 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).
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel - SPI bus.

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.
Data Words	Sets the number of data words (1 word = 8 bits) on which to trigger (one to sixteen bytes). Use the A knob to change the value. Available when Trigger On = **Data**.
Data	Sets the data pattern on which to trigger. The number of bits shown depends on the **Data Words** setting. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the 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).
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Bus Trigger Settings panel - USB bus.

Field or control	Description
Source	Select the **USB** bus on which to trigger.
Trigger On	Select the type of information on which to trigger.
Handshake Type	Sets the handshake packet type on which to trigger. Available when Trigger On = Handshake Packet.
Packet Type	Sets the special packet type on which to trigger. Available when Trigger On = Special Packet.
Error Type	Sets the error type on which to trigger. Available when Trigger On = Error.
Address	Sets the token packet address pattern on which to trigger. Use in conjunction with the **Trigger When** field to specify the exact trigger condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = Token Packet and Token Type = all except SOF (0101).
Address Low, Address High	Sets the boundary address conditions when testing for in-range or out of range conditions. Available when Trigger On = Token Packet and Trigger When = Inside Range or Outside Range.
Data Low, Data High	Sets the boundary data conditions when testing for in-range or out of range conditions. Available when Trigger On = Data Packet and Trigger When = Inside Range or Outside Range.
Token Type	Sets the token packet type on which to trigger. Available when Trigger On = Token Packet.
Endpoint	Sets the token packet endpoint pattern on which to trigger. Use in conjunction with the **Trigger When** field to specify the exact trigger condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = Token Packet and Token Type = all except SOF (0101).
Frame Number	Sets the frame number pattern on which to trigger. Use in conjunction with the **Trigger When** field to specify the exact trigger condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the A and B knobs to select and change the values. Available when Trigger On = Token Packet and Token Type = SOF (0101).
Data Packet Type	Sets the data packet type on which to trigger. Available when Trigger On = Data Packet.

Field or control	Description
Data Bytes	Sets the number of data bytes on which to trigger (one to two bytes). Tap the field and use the A knob to change the value. Available when Trigger On = Data Packet.
Byte Offset	Sets the byte offset (**Don't Care** or the number of bytes). Tap the field and use the A knob to change the value. Available when Trigger On = Data Packet.
Data	Sets the data packet pattern on which to trigger. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the Trigger When field to specify the exact trigger condition. Tap the **Binary**, **Hex**, or **ASCII** field and use the A and B knobs to select and change the values. Available when Trigger On = Data Packet and Trigger When = anything but Inside Range or Outside Range.
Trigger When	Sets the trigger when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified trigger type. Tap the Binary or Hex field and use the A and B knobs to select and change the values. Available when Trigger On = Token Packet or Data Packet.
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).
Mode & Hold off panel	See *Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls* on page 304.

Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Edge Trigger menu*
- *Logic Trigger*
- *Pulse Width Trigger menu*
- *Rise/Fall Time Trigger menu*
- *Runt Trigger menu*
- *Sequence Trigger menu*
- *Setup & Hold Trigger menu*
- *Timeout Trigger menu*
- *Window Trigger menu*

Edge Trigger configuration menu

Use the Edge Trigger menu to trigger the oscilloscope 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**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control. The MSO58LP also has an AUX edge trigger source, which sets the instrument to use a trigger signal connected to the AUX front-panel connector.
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 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, falling, or either direction).

Mode & Holdoff panel (Edge Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Bus Trigger Menu*
- *Logic Trigger menu*
- *Pulse Width Trigger menu*
- *Rise Fall Time Trigger menu*
- *Runt Trigger menu*
- *Sequence Trigger menu*
- *Setup and Hold Trigger menu*
- *Timeout Trigger menu*
- *Window Trigger menu*

Logic Trigger configuration menu

Use the Logic trigger to trigger the oscilloscope when the specified logic conditions occur on any combination of analog and digital inputs. The logic conditions include the state of each input, the condition to test (inputs go true, false, or are within a time limit), and the Boolean function of the inputs.

To open the Logic Trigger menu:

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

Settings panel (Logic Trigger configuration menu) - fields and controls.

Field or control	Description
Use Clock Edge?	Enables or disables finding logic patterns that occur on the specified clock edge.
Logic Pattern Define Inputs	Opens the **Logic Trigger - Define Inputs** menu where you define the logic state (**High**, **Low**, or **Don't Care**), and the signal threshold level that defines the logic state (high or low), for each analog or digital signal. See *Logic Trigger - Define Inputs configuration menu* on page 310.
Trigger When (Use Clock Edge = No)	Defines the waveform condition on which to trigger. ■ **Goes True**: All conditions change to a true state. ■ **Goes False**: All conditions change to a false state. ■ **Is True > Limit**: Condition remains true longer than a specified time. ■ **Is True < Limit**: Condition remains true for less than a specified time. ■ **Is True = Limit**: Condition remains true for a specified time (within ± 5%). ■ **Is True ≠ Limit**: Condition does not remain true for a specified time (within ± 5%).
Clock Source (Use Clock Edge = Yes)	Sets the signal to use as the clock. The clock signal can be a digital, analog, or math waveform
Clock Edge (Use Clock Edge = Yes)	Sets the signal transition edge (rising, falling, or either) for evaluating the logic condition at that clock transition.
Clock Threshold (Use Clock Edge = Yes)	Sets the threshold level that the clock signal must pass through to be considered a valid transition. The clock threshold value is independent of the input signal threshold(s).
Define Logic	Sets the logic condition that must occur with all inputs to cause a trigger event. Sets the logic condition that must occur with all inputs. ■ **AND**: All conditions are true. ■ **OR**: Any condition is true. ■ **NAND**: One or more conditions are true. ■ **NOR**: No conditions are true.

Mode & Holdoff panel (Logic Trigger configuration menu) - fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Bus Trigger Menu*
- *Edge Trigger menu*
- *Pulse Width Trigger menu*
- *Rise Fall Time Trigger menu*
- *Runt Trigger menu*
- *Sequence Trigger menu*
- *Setup and Hold Trigger menu*
- *Timeout Trigger menu*
- *Window Trigger menu*

Logic Trigger - Define Inputs configuration menu

Use this menu to set the signal sources, logic states, and threshold levels to use for the Logic trigger.

To open the Logic Trigger - Define Inputs configuration menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set **Trigger Type** to **Logic** (if it is not already set to this).
3. Tap the **Logic Pattern Define Inputs** button.

Logic Trigger - Define Inputs configuration menu fields and controls.

Field or control	Description
Chx (analog channels) or Dx (digital channels)	Use to select the signal sources logic condition on which to perform the logic search (**High**, **Low**, **Don't Care**). Tap to select. If a channel is a digital channel, tap the + symbol to open the list of digital inputs (D0-D7) from which to select individual logic conditions for the digital signals. Use the **Threshold** field to set the signal level that must be exceeded for that signal to be true (logical 1).
Set All	Sets all signal sources to detect a logic **High**, **Low**, or **Don't Care** condition.

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. Pulse Width triggers are often used to troubleshoot digital signals.

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 or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Trigger When	■ **< Limit**: A pulse width is less than the specified time limit. ■ **> Limit**: A pulse width is greater than the specified time limit. ■ **= Limit**: A pulse width is equal to the specified time limit (±5%). ■ **≠ Limit**: A pulse width does not equal (is greater than or less than) the specified time limit (±5%). ■ **Inside Range**: A pulse width is in the specified time range (±5%). ■ **Outside Range**: A pulse width is outside of the specified time range (±5%).
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 or Outside Range)	Sets the longest acceptable pulse width time period for the range condition.
Low Time Limit (Trigger When = Inside Range or Outside Range)	Sets the shortest acceptable pulse width time period for the range condition.
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Logic Qualification	Set to On to enable logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the Logic Qualification - Define Inputs dialog. Use this dialog to set the logic state, threshold levels, and logic operation of the input signals. See *Logic Qualification - Define Inputs configuration menu* on page 327. Available when Logic Qualification = On.

Mode & Holdoff panel (Pulse Width Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Bus Trigger Menu*
- *Edge Trigger menu*
- *Logic Trigger menu*
- *Rise Fall Time Trigger menu*
- *Runt Trigger menu*
- *Sequence Trigger menu*
- *Setup and Hold Trigger menu*
- *Timeout Trigger menu*
- *Window 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. Types that require multiple inputs will replace this control with a different source definition control.
Trigger When	■ **< Limit**: A signal has a rise time less than the specified time limit. ■ **> Limit**: A signal has a rise time greater than the specified time limit. ■ **= Limit**: A signal has a rise time that is equal to the specified time limit (±5%). ■ **≠ Limit**: A signal has a rise time that does not equal (is greater than or less than) the specified time limit (±5%).
Time Limit	Sets the time period condition to be met.
Slope	Sets the signal transition direction to detect. (rising, falling, or either direction).
Upper Threshold	Sets the upper threshold amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower threshold amplitude level through which the signal must pass to be considered a valid transition.
Logic Qualification	When set to On enables logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the **Logic Qualification - Define Inputs** configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See *Logic Qualification - Define Inputs configuration menu* on page 327. Available when Logic Qualification = On.

Mode & Holdoff panel (Rise/Fall Time Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Bus Trigger Menu*
- *Edge Trigger menu*
- *Logic Trigger menu*
- *Pulse Width Trigger menu*
- *Runt Trigger menu*
- *Sequence Trigger menu*
- *Setup and Hold Trigger menu*
- *Timeout Trigger menu*
- *Window 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 or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
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. ■ **= Limit**: A runt signal event occurs that has a pulse width that is equal to the specified time limit (±5%). ■ **≠ Limit**: A runt signal event occurs that has a pulse width that does not equal (is greater than or less than) the specified time limit (±5%).
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 threshold amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower threshold amplitude level through which the signal must pass to be considered a valid transition.
Logic Qualification	Set to On to enable logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the **Logic Qualification - Define Inputs** configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See *Logic Qualification - Define Inputs configuration menu* on page 327. Available when Logic Qualification = On.

Mode & Holdoff panel (Runt Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Bus Trigger Menu*
- *Edge Trigger menu*
- *Logic Trigger menu*
- *Pulse Width Trigger menu*
- *Rise Fall Time Trigger menu*
- *Sequence Trigger menu*
- *Setup and Hold Trigger menu*
- *Timeout Trigger menu*
- *Window Trigger menu*

Sequence Trigger configuration menu

Use Sequence triggers to trigger on a second (B) event after a first (A) event occurs. You can specify to trigger on the first occurrence of event B (with or without a time delay), or trigger after a specified number of B events occur.

To open the Sequence trigger configuration menu:

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

Settings panel (Sequence Trigger configuration menu) fields and controls.

Field or control	Description
A Trigger Event	Tap to open the **A Trigger Event** menu to select the first (A) event trigger condition. See *A Trigger Event configuration menu* on page 321. If the A event does not occur, no trigger event is generated. The **Sequence** trigger type is not available in the A Trigger Event menu.
B Trigger Event	Tap to open the **B Trigger Event** menu to select the second (B) event trigger condition. See *B Trigger Event configuration menu* on page 321. If the A event occurs but the B event does not occur, no trigger event is generated. The **Sequence** trigger type is not available in the B Trigger Event menu.
After the A Trigger Event is found: Trigger on the 1st B event	Sets the oscilloscope to trigger on the first occurrence of the B event trigger conditions.
After a Delay of:	Sets a time delay condition for the Trigger on 1st B event condition. The oscilloscope waits the specified time period after the A event before detecting and triggering on the B event condition. Available when After the A Trigger Event is found = Trigger on the 1st B event.
After the A Trigger Event is found: Trigger on the Nth B event	Sets the B trigger event to wait for a specified number of trigger events before generating a trigger.
Where N is:	Sets the number of B trigger events that must occur before triggering the oscilloscope. Available when After the A Trigger Event is found = Trigger on the Nth B event.

Mode & Holdoff panel (Sequence Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Holdoff is not compatible with Sequence triggering.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Bus Trigger Menu*
- *Edge Trigger menu*
- *Logic Trigger menu*
- *Pulse Width Trigger menu*
- *Rise Fall Time Trigger menu*
- *Runt Trigger menu*
- *Setup and Hold Trigger menu*
- *Timeout Trigger menu*
- *Window Trigger menu*

A Trigger Event configuration menu

Use this menu to set the trigger conditions for the A trigger event of a Sequence trigger.

To open the A Trigger Event menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Sequence**.
3. Tap the **A Trigger Event** button.

A Trigger Event menu (Sequence Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Type	Select the A event trigger type.
Other fields and controls	The fields and controls shown depend on the selected Trigger Type. Use the following links to access setting information on a specific trigger. ■ *Bus Trigger Menu* ■ *Edge Trigger menu* ■ *Logic Trigger* ■ *Pulse Width Trigger menu* ■ *Rise/Fall Time Trigger menu* ■ *Runt Trigger menu* ■ *Sequence Trigger menu* ■ *Setup & Hold Trigger menu* ■ *Timeout Trigger menu* ■ *Window Trigger menu* ***NOTE.*** *The Sequence Trigger type is not available in the A Trigger Event menu.*

B Trigger Event configuration menu

Use this menu to set the trigger conditions for the B trigger event of a Sequence trigger.

To open the B-Trigger Event configuration menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Sequence**.
3. Tap the **B Trigger Event** button.

B Trigger Event menu (Sequence Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Type	Select the B event trigger type.
Other fields and controls	The fields and controls shown depend on the selected Trigger Type. Use the following links to access setting information on a specific trigger. ■ *Bus Trigger Menu* ■ *Edge Trigger menu* ■ *Logic Trigger* ■ *Pulse Width Trigger menu* ■ *Rise/Fall Time Trigger menu* ■ *Runt Trigger menu* ■ *Sequence Trigger menu* ■ *Setup & Hold Trigger menu* ■ *Timeout Trigger menu* ■ *Window Trigger menu* ***NOTE.*** *The Sequence Trigger type is not available in the A Trigger Event menu.*

Setup and Hold Trigger configuration menu

Use the Setup & Hold trigger to trigger on a waveform when a data signal changes state inside of a specified setup and hold time, relative to a clock edge.

To open the Setup & Hold trigger configuration menu:

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

Settings panel (Setup & Hold Trigger configuration menu) fields and controls.

Field or control	Description
Clock Source	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform.
Clock Level	Sets the threshold level that the clock signal must pass through to be considered a valid transition. The clock threshold value is independent of the input signal threshold(s).
Clock Edge	Sets the polarity of the clock edge (rising or falling) for evaluating the other menu conditions. The Logic menu also lets you set the clock edge to either edge.
Data Sources: Define Inputs	Opens the Setup & Hold Trigger - Define Inputs menu. Use this menu to select the input signals and their thresholds. See *Setup and Hold Trigger - Define Inputs configuration menu* on page 324.
Setup Time	Sets the length of time that data signal should be stable and not change before a clock edge occurs.
Hold Time	Sets the length of time that data signal should be stable and not change after a clock edge occurs.

Mode & Holdoff panel (Setup & Hold Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Bus Trigger Menu*
- *Edge Trigger menu*
- *Logic Trigger menu*
- *Pulse Width Trigger menu*
- *Rise Fall Time Trigger menu*
- *Runt Trigger menu*
- *Sequence Trigger menu*
- *Timeout Trigger menu*
- *Window Trigger menu*

Setup and Hold Trigger - Define Inputs configuration menu

Use this menu to set the input signals and their threshold levels for the Setup & Hold trigger.

To open the Setup & Hold Trigger - Define Inputs dialog:

1. Double-tap the **Trigger** Badge.
2. Set the **Trigger Type** to **Setup & Hold**.
3. Tap the **Data Sources Define Inputs** button.

Setup & Hold Trigger - Define Inputs configuration menu fields and controls.

Field or control	Description
Chx (analog channels) or Dx (digital channels)	Use to select the signal sources to test for the setup and hold condition. Tap to select for each input source. If a channel is a digital channel, tap the **+** symbol to open the list of digital inputs (D0-D7) from which to select individual digital signals. Use the Threshold field to set the signal level that must be exceeded for that signal to be true.
Set All	Sets all signal sources to be included or not included.

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 or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
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.
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.
Logic Qualification	When set to On, enables logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the **Logic Qualification - Define Inputs** configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See *Logic Qualification - Define Inputs configuration menu* on page 327. Available when Logic Qualification = On.

Mode & Holdoff panel (Timeout Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Bus Trigger Menu*
- *Edge Trigger menu*
- *Logic Trigger menu*
- *Pulse Width Trigger menu*
- *Rise Fall Time Trigger menu*
- *Runt Trigger menu*
- *Sequence Trigger menu*
- *Setup and Hold Trigger menu*
- *Window Trigger menu*

Logic Qualification - Define Inputs configuration menu

Use this menu to set the logic state, and threshold levels to use for the trigger.

This menu is available when the trigger type is set to Pulse Width, Timeout, Runt, Window or Rise / Fall Time.

To open the Logic Qualification - Define Inputs menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Pulse Width**, **Timeout**, **Runt**, **Window** or **Rise / Fall Time**.
3. Tap Logic Qualification to **On.**
4. Tap the **Define Inputs** button.

Logic Qualification - Define Inputs (Trigger configuration menu) fields and controls.

Field or control	Description
Chx (analog channels) or Dx (digital channels)	Use to select the signal source logic condition on which to perform the logic trigger (High, Low, Don't Care). Tap to select for each input source. If a channel is a digital channel, tap the + symbol to open the list of digital inputs (D0-D7) from which to select logic conditions for the individual digital signals. Use the Threshold field to set the signal level that must be exceeded for that signal to be true (logical 1).
Set All	Sets all signal sources to detect a logic **High**, **Low**, or **Don't Care** condition.
Define Logic	Sets the logic condition that must occur with all inputs. ▪ **AND**: All conditions are true. ▪ **OR**: Any condition is true. ▪ **NAND**: One or more conditions are true. ▪ **NOR**: No conditions are true.

Window Trigger configuration menu

Use the Window Trigger to trigger when a signal rises above an upper threshold level or falls below a lower threshold level (the 'window'), with or without a time limit constraint.

To open the Window trigger menu:

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

Settings panel (Window Trigger configuration menu) fields and controls.

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Trigger When	■ **Enters Window**: The signal outside a window enters the window defined by the upper and lower threshold settings. ■ **Exits Window**: The signal exits the window defined by the upper and lower threshold settings. ■ **Inside > Limit**: The signal remains inside the window longer then the specified time limit. ■ **Outside > Limit**: The signal remains outside the window longer then the specified time limit.
Upper Threshold	Sets the amplitude threshold value for the upper edge of the window.
Lower Threshold	Sets the amplitude threshold value for the lower edge of the window.
Time Limit	Sets the time period condition to be met. Available when Trigger When = Inside > Limit or Outside > Limit.
Threshold Crossing (Trigger When = Outside > Limit)	■ **Upper**: A signal remains above the upper threshold level for longer than the specified time limit before crossing the upper threshold level to a lower level. ■ **Lower**: A signal remains below the lower threshold level for longer than the specified time limit before crossing the lower threshold level to a higher level. ■ **Either**: A signal remains outside (above or below) the two threshold levels for longer than the specified time limit before crossing either threshold level. ■ **None**: A signal remains outside the two specified threshold levels for longer than a specified time limit.
Threshold Crossing (Trigger When = Inside > Limit)	■ **Upper**: A signal remains between two thresholds for longer than the specified time limit before crossing through the upper threshold. ■ **Lower**: A signal remains between two thresholds for longer than the specified time limit before crossing through the lower threshold. ■ **Either**: A signal remains between two thresholds for longer than the specified time limit before crossing through either the upper or lower threshold. ■ **None**: A signal remains between two threshold levels for longer than a specified time limit.
Logic Qualification	When set to On, enables logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the **Logic Qualification - Define Inputs** configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See *Logic Qualification - Define Inputs configuration menu* on page 327. Available when Logic Qualification = On.

Mode & Holdoff panel (Window Trigger configuration menu) fields and controls.

Field or control	Description
Trigger Mode	The trigger mode determines 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 even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT. Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.

Other trigger types.

- *Bus Trigger Menu*
- *Edge Trigger menu*
- *Logic Trigger menu*
- *Pulse Width Trigger menu*
- *Rise Fall Time Trigger menu*
- *Runt Trigger menu*
- *Sequence Trigger menu*
- *Setup and Hold Trigger menu*
- *Timeout Trigger menu*

Virtual Keyboard

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

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. Double-tap to select individual words. Triple-tap to select all text in the field.

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

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
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 to set display mode, 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. **Stacked** mode draws each waveform in an individual section, or slice, of the screen, stacked one on top of the other. This lets you view each waveform uncluttered by overlaying waveforms. Each waveform slice displays its own vertical scale units. A trigger level indicator in a slice indicates that that waveform is the trigger source.
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.
Waveform Intensity	Sets the brightness of the waveform. Tap the field and use the A knob to set the waveform intensity.

Fields or controls	Description
Graticule Style	Sets the type of graticule to display. **Grid** shows a traditional grid on the instrument display. This is the default grid mode. **Time** displays vertical graticules for just the horizontal (time) scale units. **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 concepts

Acquisition concepts

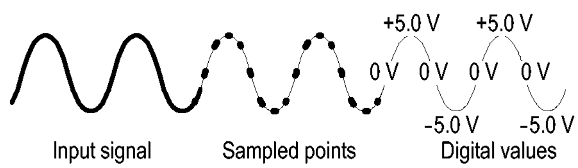
The Acquisition system sets which data points are used to acquire waveforms.

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.

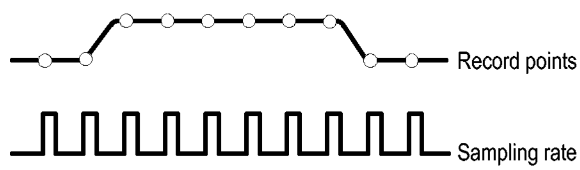
Sampling process

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.



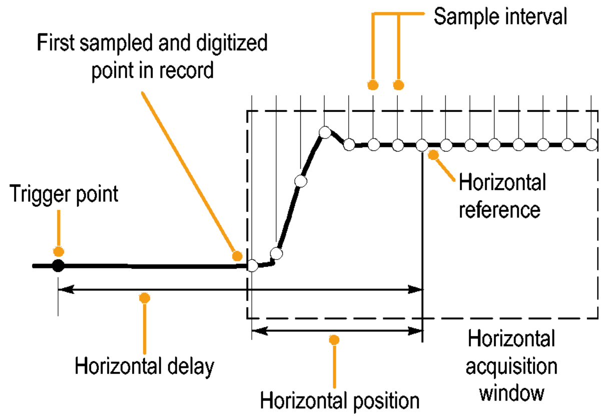
Interpolated Real-Time sampling

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

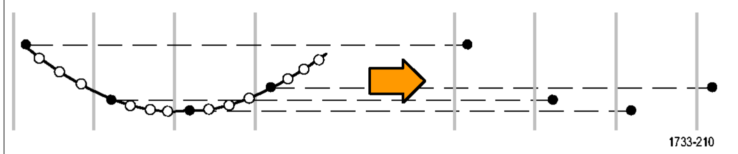
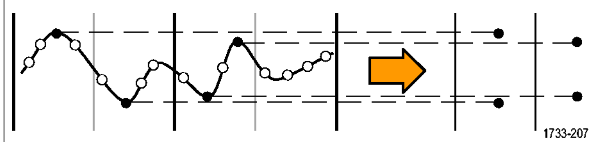
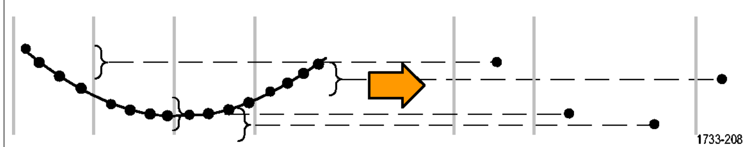
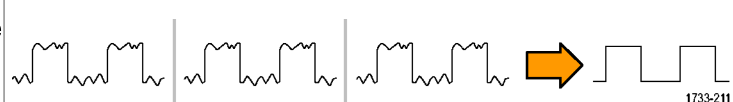
Your instrument can interpolate between the samples it acquires when it does not have all of the actual samples it needs to fill the waveform record. Linear interpolation computes record points between actual acquired samples by using a straight line fit.

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.	
Peak Detect mode retains the highest and lowest values of all the samples in two consecutive acquisition intervals. This mode only works with real-time, noninterpolated sampling and is useful for catching high frequency glitches.	
High Res mode applies unique FIR filtering based on the current sample rate. This FIR filter maintains maximum bandwidth possible for that sample rate while rejecting aliasing. The filter removes noise from the oscilloscope amplifiers and ADC above the usable bandwidth for the selected sample rate. Implementation of the filter in hardware, ahead of the trigger and storage, reduces trigger jitter and enables Fast Acq while in High Res mode.	
Envelope mode finds the highest and lowest record points over many acquisitions. Envelope uses Peak Detect for each individual acquisition.	
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.	
FastAcq™ mode is helpful in finding elusive signal anomalies. Fast acquisition mode reduces the dead time between waveform acquisitions, enabling the capture and display of transient events such as glitches and runt pulses. Fast acquisition mode can also display waveform phenomena at an intensity that reflects their rate of occurrence.	

Acquisition mode	
FastFrame™ segmented memory acquisition uses multiple trigger events to capture widely spaced events of interest at high sample rates while conserving acquisition memory. Capturing thousands of frames is possible, allowing analysis of long-term trends and changes in the bursting signal.	
Roll Mode scrolls sequential waveform points across the display in a right-to-left rolling motion. Works at sample rates up to **10** MS/s with a maximum record length of **??** MS. Roll mode starts automatically when the timebase is set to ≥40 ms/div.	

FastFrame concepts

FastFrame™ lets you to capture only the waveform, or waveform segment of interest, eliminating the dead space between conditions of interest. Each captured event is stored in its own numbered memory segment. Multiple memory segments, or frames, can then be viewed individually in the order they were captured, or layered to show their similarity and contrast.

Advantages of using FastFrame include:

- The high waveform capture rate increases the probability of capturing infrequent events
- The waveform detail is preserved by using high sample rates
- The events are captured while ignoring the dead time between them, ensuring efficient use of the record length memory
- The segments can be quickly and visually compared to determine if an anomaly “sticks out” of the overlaid stack

Each frame can be viewed individually and you can scroll through them by selecting frame numbers with a mouse, virtual keypad or the multi-purpose knob on the instrument’s main console. When a particular frame of interest is identified, you can use the instrument’s features to characterize, measure, zoom and analyze the waveform in detail.

To quickly see anomalies that stand out from the common shape of the waveform, multiple frames can be overlaid to show common and outlying points. Enabling Overlay Frames in the FastFrame configuration menu overlays all frames in the current acquisition using color to highlight how frequently the points are overlaid on each other.

You can also view multiple signal input frames simultaneously by using stacked waveform display mode. When you scroll through the frames, the oscilloscope displays the acquisition frame for all waveforms captured.

Debugging signal errors with time stamps

FastFrame mode provides a different type of functionality for digital designers. For example, if your processor system is being infrequently interrupted, it can be difficult to gather timing information with an oscilloscope. If you don’t know when or how frequently the event occurs, you can’t set up the instrument in normal acquisition mode and be assured of capturing the information you need.

FastFrame mode can do this easily by providing information on the interval between frames. For example, the active high interrupt strobe on a microprocessor system is measured to be roughly 100 nanoseconds wide, so we set up the oscilloscope to capture 100 frames of 1250 points. In this example, the shape of the pulse is not of particular interest. We are, however, interested in the time of the pulses’ rising edges.

After turning FastFrame on and selecting “Single Acquisition” to capture 100 frames, you use the readouts in the FastFrame results badge to compare the time stamp data at the trigger point. The “Reference Frame” was chosen to be the first interrupt pulse and the “Selected Frame” is the fourth pulse. The time difference between these pulses is shown in the Delta readout on the badge.

The time stamps of all the frames can be output in tabular form for in-depth analysis using Excel or a wide variety of other popular software tools that read comma-delimited files (.CSV).

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 three 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.
- Auto interpolation. Select the best interpolation method.

Coupling

All instruments and probes specify a maximum signal level. Do not exceed the limit, even momentarily, as the input channel or probe may be damaged. Use external attenuators if necessary to prevent exceeding the limits.

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).

The input resistance of each input channel can be 1 MΩ or 50 Ω. To properly terminate signals when using coaxial cables, or to support active probes with different termination requirements, select the termination in the Channel menu Vertical Settings panel.

All probes expect a specific coupling and input termination. Both coupling and input termination are displayed on the screen. If the instrument determines the coupling and termination required by the probe, either implicitly because of the TekProbe/TekVPI interface or through performing a probe compensation, the instrument sets the required coupling and input termination.

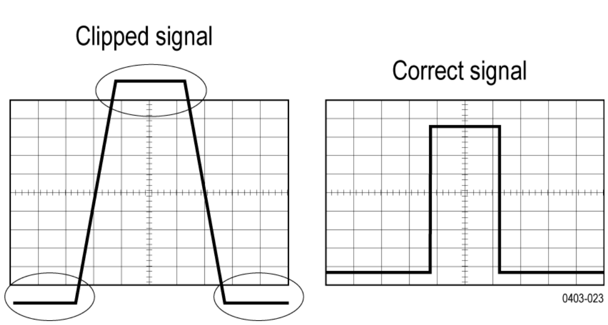
Consider the following when you use 50 Ω termination with any channel:

- The instrument does not accurately display frequencies under 200 kHz if AC coupling is selected.
- The instrument reduces the maximum volts per division setting for the channel, since input amplitudes appropriate for the higher settings would overload the 50 Ω input.

Scaling and positioning

The scaling and positioning controls determine the portion of the input signal received by the acquisition system. Set 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* on page 340.



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* on page 341 .

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.

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 waveform display up or down relative to the acquired 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 waveform display up and down on the input signal. Offset moves the waveform display to control the portion of the waveform amplitude the display 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.

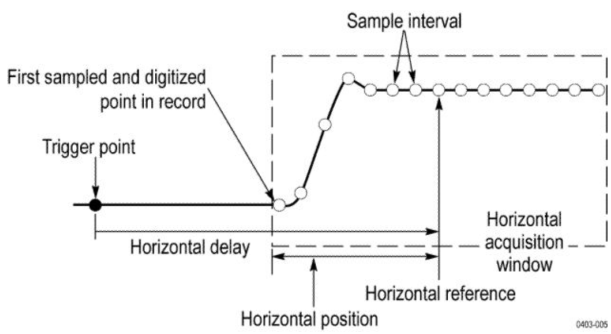
Horizontal acquisition considerations

The instrument lets you define the horizontal waveform display parameters that determine which segment of an incoming signal becomes the waveform record, following acquisition.

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.



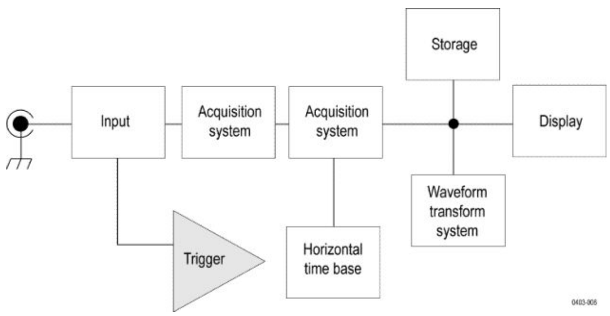
Trigger concepts

Triggering concepts

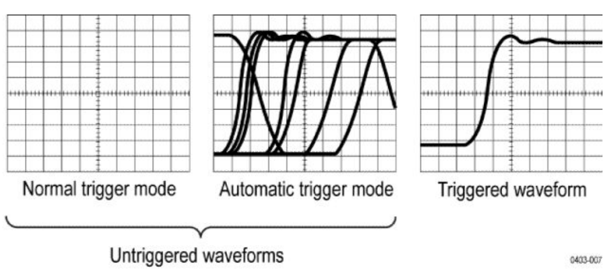
Overview

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.



Triggers help you 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 posttrigger 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.
- **Digital channels**. These sources are available if you have a digital probe connected to a FlexChannel. You can select any combination of digital channels.
- **Bus**. This source is used to trigger a parallel bus or a serial bus. You can include any combination of analog, math, or digital channels to build a parallel bus, or use any channel as a component in a serial bus.

Trigger types

The available trigger types include:

Edge. This is the simplest and most commonly used trigger type, used with both analog and digital signals. An edge trigger event occurs when the trigger source passes through a specified voltage level in the specified direction (rising or falling signal voltage).

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

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

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. Can detect positive or negative runts, or only those wider than a specified width. These pulses can also be qualified by the logical state of other channels.

Window. Use the Window trigger to trigger when the input signal rises above an upper threshold level or falls below a lower threshold level. Trigger the instrument as the signal is entering or leaving the threshold window. Qualify the trigger event in terms of time by using the Trigger When Wider option, or by the logical state of other channels using the Trigger When Logic option.

Logic. These are special-purpose triggers that are primarily used with digital logic signals. Logic triggers are available on the main and B event triggers.

Setup & Hold. Trigger when a logic input changes state inside the setup and hold times relative to the clock. This type triggers on a setup and hold violation.

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.

Sequence. Use the A Trigger Event with the B Trigger Event to capture complex data.

Bus. This trigger is used with both analog and digital signals to set up parallel buses or serial buses. A bus trigger event occurs when the instrument detects a bus pattern that you specify for a parallel bus, or a bus cycle you select for a serial bus. A bus is defined in a bus menu.

- Parallel. Use to trigger on Parallel bus signals.
- I^{2}C. Use to trigger on Inter-IC signals: start, stop, repeated start, missing acknowledge, address, data, and address and data.
- SPI. Use to trigger on Serial Peripheral Interface signals.
- RS-232. Use to trigger on RS-232 signals.

- CAN. Use to trigger on CAN Bus signals.
- LIN. Use to trigger on LIN bus signals.
- FlexRay. Use to trigger on FlexRay bus signals.
- USB. Use to trigger on USB bus signals.
- Ethernet. Use to trigger on Ethernet bus signals.
- Audio. Use to trigger on Audio bus signals.

Trigger modes

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

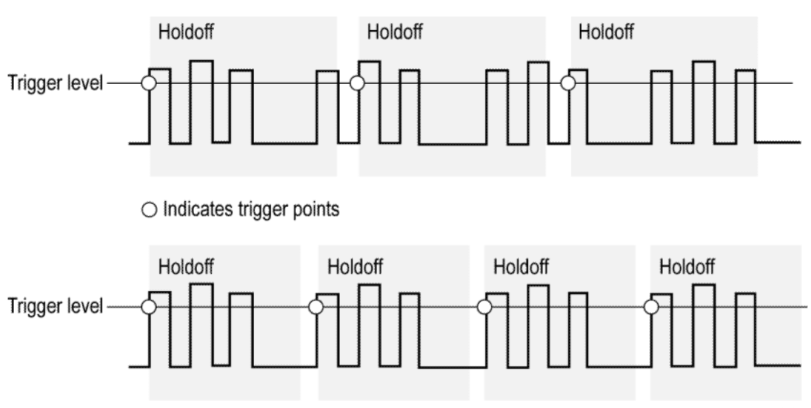
- **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the instrument does not acquire a waveform, and the last waveform record acquired remains "frozen" on the display. If no last waveform exists, no waveform is displayed.
- **Auto** trigger 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; therefore, the waveform will appear to roll across the screen. If valid triggers occur, the display will become stable.

Trigger holdoff

Trigger holdoff can help stabilize triggering. When the instrument recognizes a trigger event, it disables the trigger system until acquisition is complete. In addition, the trigger system remains disabled during the holdoff period that follows each acquisition. Adjust holdoff to obtain stable triggering when the instrument is triggering on undesired trigger events.

A digital pulse train is a good example of a complex waveform. Each pulse looks like any other, so many possible trigger points exist. Not all of these will result in the same display. The holdoff period allows the instrument to trigger on the correct edge, resulting in a stable display.



At the longer holdoff time for the top waveform, unstable triggering occurs. With a shorter holdoff set for the bottom waveform, triggers all occur on the first pulse in the burst to remedy the unstable trigger.

The Holdoff setting range is 0 s (minimum holdoff available) to 10 s (maximum holdoff available). For more information on how to set holdoff, see *Set Trigger Holdoff* on page 103. If you select Auto holdoff, the instrument selects a holdoff value for you. When Trigger Holdoff is set to Random, the instrument delays the trigger a random amount of time between triggers. This means that successive acquisitions are unrelated to the previous trigger signal.

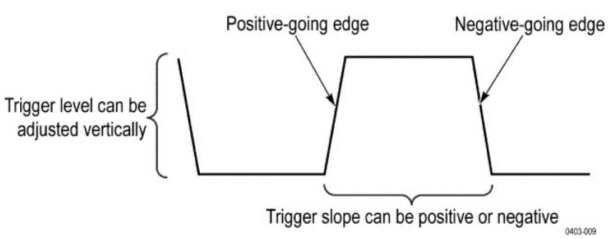
Trigger coupling

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 Rej**. This coupling attenuates signals below 50 kHz before passing the signal to the trigger circuitry.
- **Noise Rej**. 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.

Trigger delay

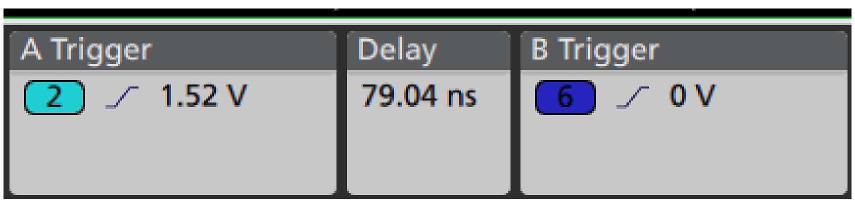
Use the Trigger Delay to trigger the instrument a specified period of time after the A trigger. After the A trigger arms the trigger system, the instrument triggers on the next B trigger event that occurs after the time that you specify.

You can trigger with the A trigger system alone or you can combine the A trigger with the B (Delayed) trigger to trigger on sequential events. When using sequential triggering, the A trigger event arms the trigger system, and the B trigger event triggers the instrument when the B trigger conditions are met.

A and B triggers can (and typically do) have separate sources. The B trigger condition can be based on a time delay or a specified number of counted events.

Advanced triggering

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



Bus triggering concepts

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

You can set the instrument to trigger on a parallel bus when the instrument detects a match to the bus pattern, or when the instrument detects that the value on the bus is < or > the value of the bus pattern. The pattern can be in Binary or Hex format.

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 Addr + Data bus cycle or activity.

You can set the instrument to trigger on a USB bus when the instrument detects a Sync, Reset, Suspend, Resume, End of Packet, Token (Address) Packet, Data Packet, Handshake Packet, Special Packet, or Error bus cycle or activity.

You can set the instrument to trigger on an RS232 bus when the instrument detects a Start, End of Packet, Data, or Parity Error bus cycle or activity.

You can set the instrument to trigger on an CAN bus when the instrument detects a Start of Frame, Type of Frame, Identifier, Data, Id and Data, End of Frame, Missing Acq, or Bit Stuffing Error bus cycle or activity.

You can set the instrument to trigger on an LIN bus when the instrument detects a Sync, Identifier, Data, Identifier & Data, Wakeup Frame, Sleep Frame, or Error bus cycle or activity.

You can set the instrument to trigger on an FlexRay bus when the instrument detects a Start of Frame, Indicator Bits, Frame Id, Cycle Count, Header Fields, Data, Identifier & Data, End of Frame, or Error bus cycle or activity.

You can set the instrument to trigger on an Ethernet bus when the instrument detects a Start of Frame, MAC Address, MAC Length/Type, IP Header, TCP Header, Client Data, End of Packet, Idle, or FCS (CRC) Error bus cycle or activity.

You can set the instrument to trigger on an AUDIO bus when the instrument detects a Word Select, or 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. Pulse width triggers can also be qualified by the logical state of other channels.

Timeout trigger

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

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 runt pulse, or only those wider than a specified minimum width.
- Runt pulses can also be qualified by the logical state of other channels.

Window trigger

Use the Window trigger to trigger the instrument when the input signal rises above an upper threshold level or falls below a lower threshold level.

After setting these levels, you can specify whether you want to trigger the instrument as the signal is entering or leaving the threshold window. You can further qualify the trigger event in terms of time, or by the logical state of other channels.

Logic trigger concepts

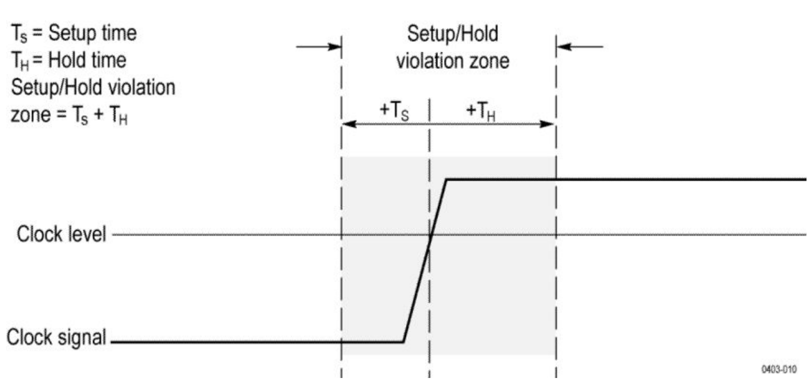
For various trigger types, touch Logic Qualification to toggle logic qualification On to trigger the instrument when the logic patterns are true. You can set each bit to be active High, Low, or Don't Care. You can also set the logic thresholds and define the logic (AND, OR, NOR, or NAND).

Setup and Hold trigger concepts

A setup/hold trigger occurs when a data signal changes state inside of the user specified setup and hold times relative to the clock. When you use setup/hold triggering, you define:

- The channel containing the logic input (the data source) and the channel containing the clock (the clock source)
- The direction of the clock edge to use
- The clocking level and data threshold that the instrument uses to determine if a clock or data transition has occurred
- The setup and hold times that together define a time range relative to the clock

Data that changes state within the setup/hold violation zone triggers the instrument. The next figure shows how the setup and hold times that you choose position the violation zone relative to the clock.



Setup/hold triggering uses the setup/hold violation zone to detect when data is unstable too near the time it is clocked. Each time trigger holdoff ends, the instrument monitors the data and clock sources. When a clock edge occurs, the instrument checks the data stream it is processing (from the data source) for transitions occurring within the setup/hold violation zone. If any occur, the instrument triggers with the trigger point located at the clock edge.

The setup/hold violation zone spans the clocking edge as shown above. The instrument detects and triggers on data that does not become stable long enough before the clock (setup time violation) or that does not stay stable long enough after the clock (hold time violation).

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. The trigger can be logic qualified.

Sequential (A B) trigger concepts

In applications that involve two or more signals, you may be able to use sequential triggering to capture more complex events. Sequential triggering uses the A (Main) trigger to arm the trigger system, and then uses the B (Delayed) trigger to trigger the instrument if a specific condition is met.

You can choose one of two trigger conditions:

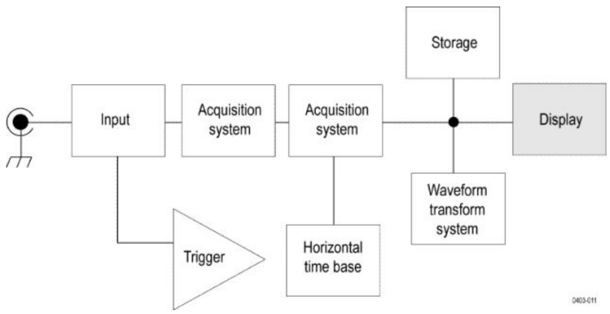
- Trigger after a Delay. After the A trigger arms the trigger system, the instrument triggers on the next B-trigger event that occurs after the trigger delay time. You can set the trigger delay time with the keypad or a multipurpose knob.
- Trigger on the Nth event. After the A trigger arms the trigger system, the instrument triggers on the Nth B event. You can set the number of B events with the keypad or a multipurpose knob.

NOTE. *The traditional delayed trigger mode called "Runs After" is controlled by the Horizontal Delay feature. You can use horizontal delay to delay acquisition from any trigger event, whether from the A trigger alone or from a sequential trigger that uses both the A and B triggers.*

Waveform display concepts

Waveform display overview

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, digital, math, reference and bus waveforms. The waveforms include channel markers, individual waveform graticule readings, and trigger source and level indicators. You can set the display to stack each waveform vertically in separate graticules, called 'slices' (the default mode), or overlay all the waveforms on the screen (traditional waveform view).

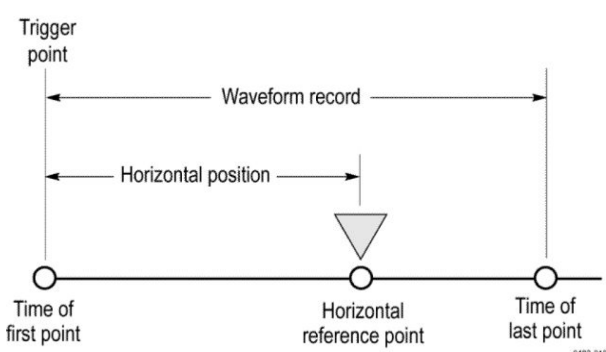
You can also add histogram, spectral, eye, and measurement results views (plots) for individual measurements. These plot views are contained within their own view window and can be moved within the overall view area.

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.



Measurement concepts

Measurement variables

By knowing how the instrument makes calculations, you may better understand how to use your instrument and how to interpret your results. The instrument uses a variety of variables in its calculations. These include:

Definition of Base and Top

Base is the value used as the 0% level in measurements such as fall time and rise time.

Top is the value used as the 100% level in measurements such as fall time and rise time. For example, if you set the 10% to 90% rise time, then the instrument calculates 10% and 90% as percentages of Top and Base, with Top representing 100%.

The exact value of Base and Top depends on which Base Top Method you select in the Reference Levels panel of a Measurement configuration menu. It also depends on if you set the reference level to be Global (applies to all measurements set as Global in the Reference Levels panel), or Local (just applies to the measurement that is set to Local).

Base, Top calculation methods

The Base Top calculation method is set in the Reference Levels panel of a Measurement configuration menu.

Auto is the default method, and automatically determines the best Base Top method to use. Most commonly sets the Base Top method to Histogram Mode.

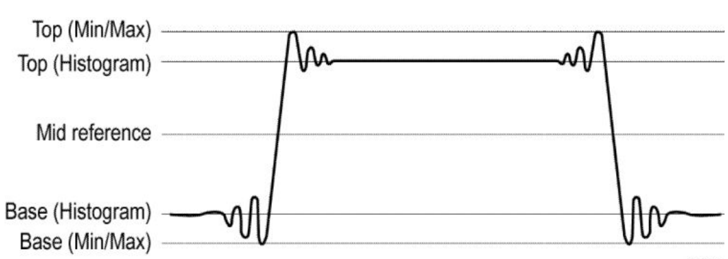
MinMax defines the 0% and the 100% waveform levels as the lowest value and the highest value samples of the waveform record. This setting is best for examining waveforms that have no large, flat portions at a common value, such as sine waves and triangle waves - almost any waveform except for pulses.

The MinMax method calculates the Top and Base values as follows:

Top = Max

and

Base = Min



Histogram Mean uses histogram analysis to calculate the mean or average value using all values above and below the waveform midpoint. Top is set to the mean high value, and Base is set to the mean low value. This setting is best for examining eye patterns and optical signals.

Histogram Mode uses histogram analysis to select the most common values either above or below the midpoint. Since this statistical approach ignores short-term aberrations (overshoot, ringing, and so on), Mode is the best setting for examining pulses.

The oscilloscope calculates the histogram-based *Top* and *Base* values as follows:

1. It makes a histogram of the record with one bin for each digitizing level.
2. It splits the histogram into two sections at the halfway point between *Min* and *Max* (also called *Mid*).
3. The level with the most points in the upper histogram is the *Top* value, and the level with the most points in the lower histogram is the *Base* value.

   If *Mid* gives the largest peak value within the histogram, the oscilloscope returns the *Mid* value for both *Top* and *Base* (this is probably a very low amplitude waveform).

   If more than one histogram level (bin) has the maximum value, the oscilloscope chooses the bin farthest from *Mid*.

This algorithm does not work well for two-level waveforms with greater than about 100% overshoot.

Histogram Eye Center uses histogram analysis of the amplitudes in the center of each bit (unit interval) while ignoring the waveform during bit transitions. The histogram sets the Top at the nominal high level and Base at the nominal low level. This is similar to the Histogram Mode, except it is less influenced by the shape of the waveform during transitions between bits.

HighRef, MidRef, LowRef

You set the various reference levels, through the Reference Levels tab of the Measure menu. They include:

High is the waveform high reference level (also HighRef). Used in all measurements. Typically set to 90%. You can set it from 0% to 100% or to a voltage level.

Mid is the waveform middle reference level (also MidRef). Mid reference levels are used in all measurements that need to find edges. Typically set to 50%. You can set it from 0% to 100% or to a voltage level.

Low is the waveform low reference level (also LowRef). Used in all measurements. Typically set to 10%. You can set it from 0% to 100% or to a voltage level.

High, mid and low reference levels can be set uniquely for each measurement source. Reference levels can also be set differently for rising edge detection and falling edge detection.

Other variables

The instrument also measures several values itself that it uses to help calculate measurements.

Record Length is the number of data points in the time base. You set it with the Horizontal menu Record Length item.

Start is the location of the start of the measurement zone (X-value). It is 0.0 samples unless you are making a gated measurement. When you use cursor gated measurements, it is the location of the left vertical cursor.

End is the location of the end of the measurement zone (X-value). It is (*RecordLength* – 1.0) samples unless you are making a gated measurement. When you use cursor gated measurements, it is the location of the right vertical cursor.

Hysteresis Is the hysteresis band of the waveform amplitude.

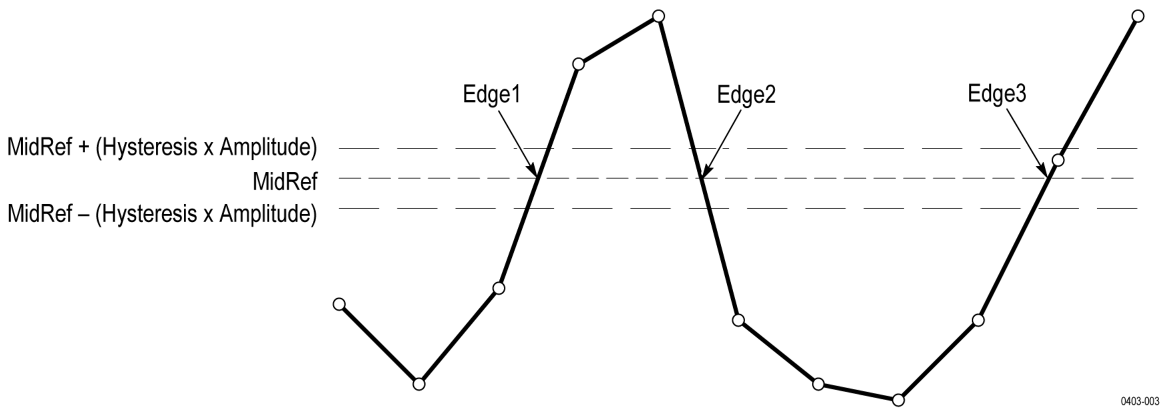
For example, once a crossing has been measured in a negative direction, the waveform data must fall below the hysteresis value of the amplitude from the *Mid* reference point before the measurement system is armed and ready for a positive crossing. Similarly, after a positive *Mid* reference crossing, waveform data must go above the hysteresis value of the amplitude before a negative crossing can be measured. Hysteresis is useful when you are measuring noisy signals, because it allows the oscilloscope to ignore minor fluctuations in the signal.

Edge calculations

Edge1, Edge2, and Edge3 refer to the first, second, and third *Mid* reference edge times, respectively.

An edge can be detected when the waveform is either rising or falling past Midref. The direction of the edges alternates, that is, if Edge1 is rising, Edge2 will be falling.

A rising edge has positive polarity. A falling edge has negative polarity.



The instrument calculates these values as follows:

1. Find the first *Mid* reference edge in the waveform record or the gated region. This is *Edge1*.
2. Continuing from *Edge1*, find the next *Mid* reference edge in the waveform record (or the gated region) of the opposite direction of *Edge1*. This is *Edge2*.
3. Continuing from Edge2, find the next *Mid* reference edge in the waveform record (or the gated region) of the same direction as Edge1. This is *Edge3*.

Cycle-cycle measurements are made on each cycle of the waveform. In the diagram above a cycle starts at Edge1 and ends at Edge3.

TPOS is the location of the sample just before the trigger point (the time reference zero sample). In other terms, it contains the domain reference location. This location is where time = 0.

TSOFF is the offset between *TPOS* and the actual trigger point. In other words, it is the trigger sample offset. Values range between 0.0 and 1.0 samples. This value is determined by the instrument when it receives a trigger. The actual zero reference (trigger) location in the measurement record is at (*TPOS* + *TSOFF*).

Missing or out-of-range samples

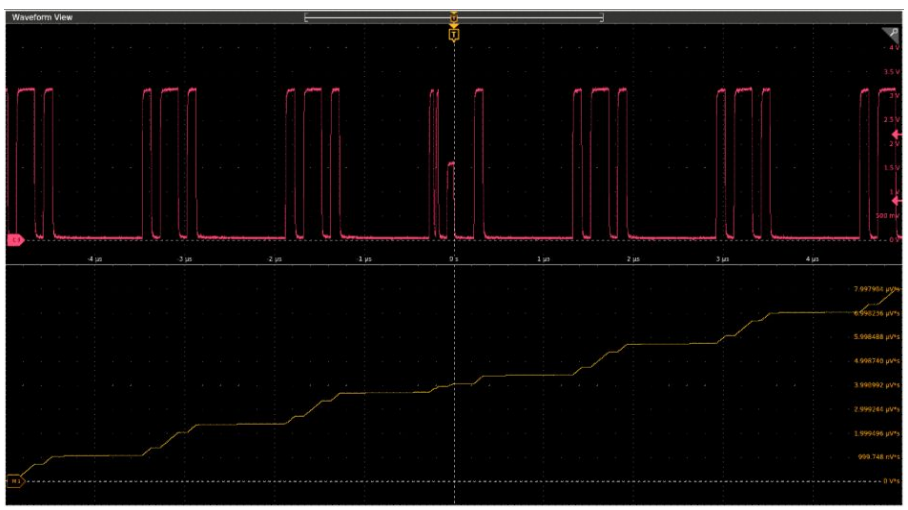
If some samples in the waveform are missing or off-scale, the measurements will interpolate between known samples to make an appropriate guess as to the sample value. Missing samples at the ends of the measurement record will be assumed to have the value of the nearest known sample. The interpolation method can be changed in User Preferences.

When samples are out of range, the measurement will give a warning to that effect (for example, CLIPPING) if the measurement could change by extending the measurement range slightly. The algorithms assume the samples recover from an overdrive condition instantaneously.

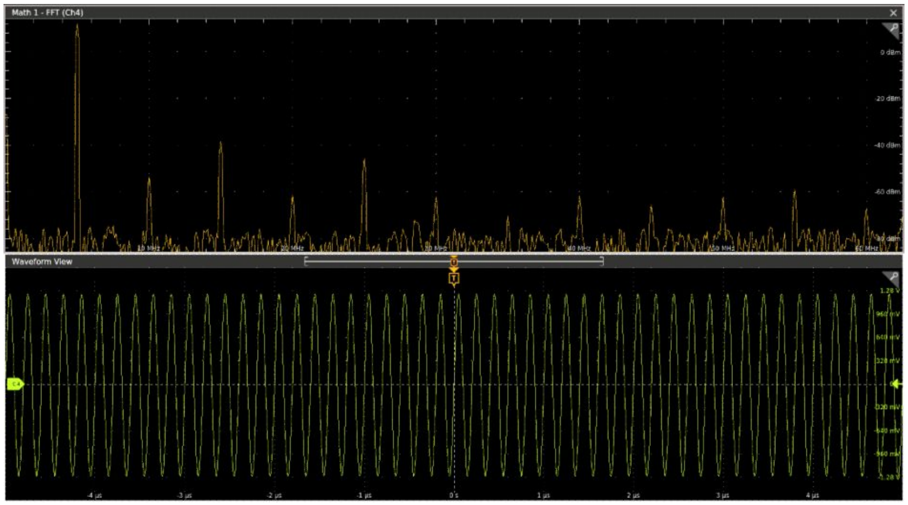
For example, if the *Mid* reference level is set directly, then *Mid* would not change even if samples were out of range. However, if *Mid* was chosen using the % choice from the **Set Levels in %** selection of the Measure menu Reference Levels tab, then *Mid* could give a CLIPPING warning.

Math waveforms

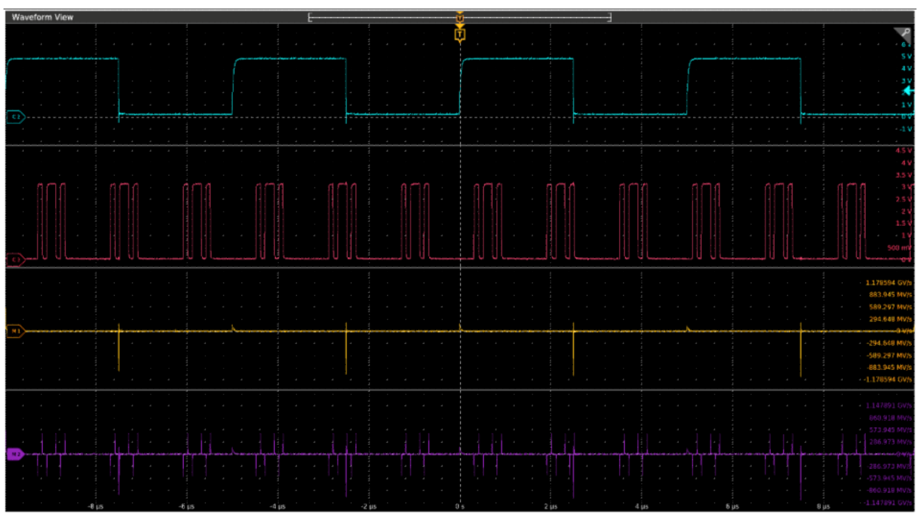
Once you have acquired waveforms or taken measurements on waveforms, the instrument can mathematically combine them to create a waveform that supports your data-analysis task. For example, you might have a waveform clouded by background noise. You can obtain a cleaner waveform by subtracting the background noise from your original waveform. Or, you can integrate a single waveform into an integral math waveform as shown below.



With spectral analysis you can analyze waveforms in the frequency domain. See the next figure.



This instrument supports mathematical combination and functional transformations of waveforms it acquires. The next figure shows this concept:



You create math waveforms to support the analysis of your channel and reference waveforms. By combining and transforming source waveforms and other data into math waveforms, you can derive the data view that your application requires. Create math waveforms that result from:

- Mathematical operations on one or several waveforms: add, subtract, multiply, and divide.
- Functional transformations of waveforms, such as integration, differentiation, and so on.
- Spectral analysis of waveforms, such as testing impulse response.

Measurement scalars can be used in math expressions. For example, you can measure the average of a waveform (using the measurement capabilities of the instrument) and subtract it from the original waveform to define a new math waveform. For example: Ch1 - mean(amplitude(Ch1)).

Math waveform elements

You can create Math waveforms from the following:

- Channel waveforms
- Reference waveforms
- Measurement scalars (automated measurements) that measure channel, reference, or math waveforms, or histograms.
- Other math waveforms
- Variables
- Filters

Dependencies

In general, math waveforms that include sources as operands are affected by updates to those sources:

- Shifts in amplitude or DC level of input sources that cause the source to clip also clip the waveform data supplied to the math waveform.
- Changes to the vertical offset setting for a channel source that clips its data also clips the waveform data supplied to the math waveform.
- Changes to the acquisition mode globally affects all input channel sources, modifying any math waveforms using them. For example, with the acquisition mode set to Envelope, a Ch1 + Ch2 math waveform will receive enveloped channel 1 and channel 2 data, and will also be an envelope waveform.
- Clearing the data in a waveform source causes a baseline (ground) to be delivered to any math waveform that includes that source until the source receives new data.

Math waveform sources

You can create Math waveforms from the following:

- Channel waveforms
- Reference waveforms
- Measurement scalars (automated measurements) that measure channel, reference, or math waveforms, or histograms.
- Other math waveforms
- Variables

Dependencies

In general, math waveforms that include sources as operands are affected by updates to those sources:

- Shifts in amplitude or DC level of input sources that cause the source to clip also clip the waveform data supplied to the math waveform.
- Changes to the vertical offset setting for a channel source that clips its data also clips the waveform data supplied to the math waveform.
- Changes to the acquisition mode globally affects all input channel sources, modifying any math waveforms using them. For example, with the acquisition mode set to Envelope, a Ch1 + Ch2 math waveform will receive enveloped channel 1 and channel 2 data, and will also be an envelope waveform.
- Clearing the data in a waveform source causes a baseline (ground) to be delivered to any math waveform that includes that source until the source receives new data.

Guidelines for working with math waveforms

Use the following guidelines when working with math waveforms:

- Keep math waveforms simple. If the math expression becomes too complex, try separating the expression into more than one math waveform and then combining the waveforms (for example, Math1 = Math2 + Math4).
- Math calculations are not available on digital channels.
- Math waveforms cannot be turned on without a math expression.
- To avoid syntax errors in a math expression, verify the use of operators, parentheses, operands, and the spelling of functions.
- If one or more reference waveforms are used in a math waveform, the record length is equal to the smallest of all the source waveforms (reference, math, or channel waveforms). The math is calculated using the first point from each source, followed by the next point, and so forth. This is true even if the sources have different times between points in the record.

Math waveform editor syntax

You can build math waveforms using the predefined expressions or the equation editor. To help you create valid math waveforms, the following tools will block most illegal entries by disabling any window element that would create an invalid entry in the math waveform expression.

Predefined expressions are accessible using the FFT or Basic Math Types..

The following syntax describes the valid math expressions you can use with the Equation Editor for the Advanced Math Type:

A math expression is composed of settings, functions, scalars and sources.

Settings have the syntax [settingName=settingValue] and are generally applied to measurements. The setting applies to everything to the right of the closing square brackets.

Example: [CoefFileName="highpass_0.25bw.flt"]HighPass(Ch1)

CoefFileName is the setting and is used as the high pass filter on channel 1.

Example: [CoefFileName="highpass_0.25bw.flt"] HighPass(Ch1) + [CoefFileName="lowpass_0.05bw.flt"] LowPass(Ch2)

The high pass filter file is applied to channel 1 and lowpass_0.05bw.flt is applied to channel 2.

Functions, except for basic and logic functions, have the syntax function(source).

In the previous examples the functions are HighPass and LowPass.

Basic and logic functions have the syntax

source1 function source2.

Examples: Ch1 * Ch2

Ch1 AND Ch2

Ch1 >= Ch2

Logic functions, ==|<|>|!=|<=|>=|AND|OR|NAND|NOR|XOR|EQV result in a waveform consisting of binary 0 and 1 values.

Scalars can be integers, floating point values, PI or meas<x>.

Sources can be Ch<x>, Ref<x>, Math<x>

Math waveform differentiation

The math capabilities of the instrument include waveform differentiation. This allows you to display a derivative math waveform that indicates the instantaneous rate of change of the waveform acquired.

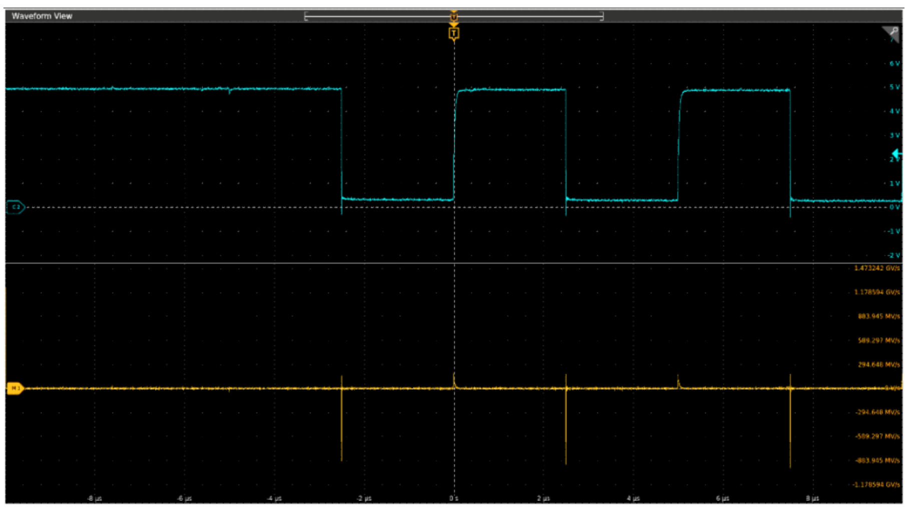
Derivative waveforms are used in the measurement of slew rate of amplifiers and in educational applications. You can create a derivative math waveform and then use it as a source for another derivative waveform. The result is the second derivative of the waveform that was first differentiated.

The math waveform, derived from the sampled waveform, is computed based on the following equation:

$Yn = (X(n + 1) - Xn) * 1/T$

Where: **X** is the source waveform, **Y** is the derivative math waveform, and **T** is the time between samples.

Since the resultant math waveform is a derivative waveform (see the next figure), its vertical scale is in volts/second (its horizontal scale is in seconds). The source signal is differentiated over its entire record length; therefore, the math waveform record length equals that of the source waveform.



Math waveform offset, position, and scale

The settings that you make for offset, scale, and position affect the math waveform you obtain. Here are some tips for obtaining a good display:

- Scale and position the source waveform so that it is contained on the screen. (Off-screen waveforms may be clipped, resulting in errors in the derivative waveform).
- Use vertical position and vertical offset to position your source waveform. The vertical position and offset will not affect your derivative waveform unless you position the source waveform off screen so that it is clipped.

Waveform integration

The math capabilities of the instrument include waveform integration. This allows you to display an integral math waveform that is an integrated version of the acquired waveform.

Use integral waveforms in the following applications:

- Measuring power and energy, such as in switching power supplies.
- Characterizing mechanical transducers, as when integrating the output of an accelerometer to obtain velocity.

The integral math waveform, derived from the sampled waveform, is computed based on the following equation:

$$y(n) = scale \sum_{i=1}^{n} \frac{x(i) + x(i-1)}{2} T$$

Where: **x(i)** is the source waveform, **y(n)** is a point in the integral math waveform, **scale** is the output scale factor, and **T** is the time between samples.

Since the resultant math waveform is an integral waveform, its vertical scale is in volt-seconds (its horizontal scale is in seconds). The source signal is integrated over its entire record length; therefore, the math waveform record length equals that of the source waveform.

Offset and position

When creating integrated math waveforms from live channel waveforms, consider the following:

- You should scale and position the source waveform so that it is contained on screen. (Off screen waveforms may be clipped, which will result in errors in the integral waveform.)
- You can use vertical position and vertical offset to position your source waveform. The vertical position and vertical offset will not affect your integral waveform unless you position the source waveform off screen so that it is clipped.

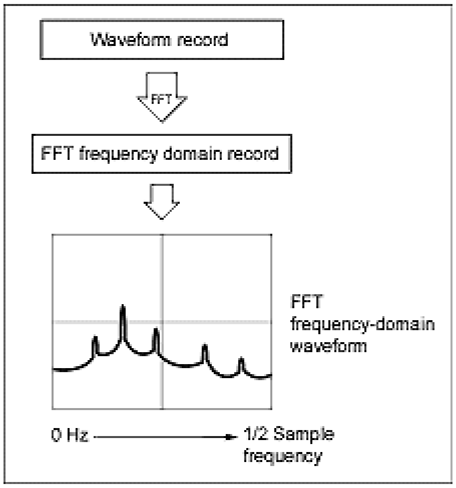
DC offset

The source waveforms that you connect to the instrument often have a DC offset component. The instrument integrates this offset along with the time-varying portions of your waveform. Even a few divisions of offset in the source waveform may be enough to ensure that the integral waveform saturates (clips), especially with long record lengths.

FFT process

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).



Nyquist frequency

The highest frequency that any digital oscilloscope can measure without errors is one-half of the sample rate. This frequency is called the Nyquist frequency.

The FFT waveform displays the input signal frequency components from DC (0 Hz) to the Nyquist frequency.

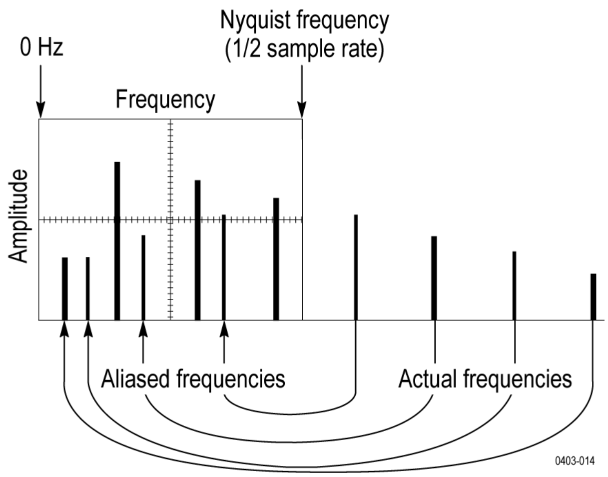
FFT and aliasing

Aliasing occurs when the input frequency of a signal is greater than one half of the sampling frequency (the sample rate).

Set the sample rate high enough so that the signals in the spectrum appear at their correct frequency as opposed to a lower aliased frequency value. Also, complex signal shapes that have many harmonics in them, such as a triangle or square wave, can appear to be OK in the time domain when in fact many of the harmonics in that signal are aliased.

One way to check for aliasing is to increase the sample rate and observe whether any of the harmonics unwrap to different frequency locations.

Another way to recognize aliasing is to realize that higher order harmonics usually have decreasing magnitudes compared to lower order harmonics. Thus, if you see a series of increasing harmonic magnitude values as frequency increases then you can suspect that they may be aliased. In the spectral math waveform, the actual higher frequency components are under sampled, and therefore they appear as lower frequency aliases that "fold back" around the Nyquist point. (See the next figure.) You may test by increasing the sample rate and observing if aliases unwrap to different frequency positions.



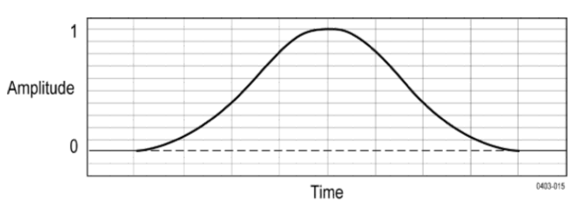
If you have a variable-frequency signal source, another way to observe aliasing is to adjust the frequency slowly while watching the spectral display. If some of the harmonics are aliased, you will see the harmonics decreasing in frequency when they should be increasing or vice versa.

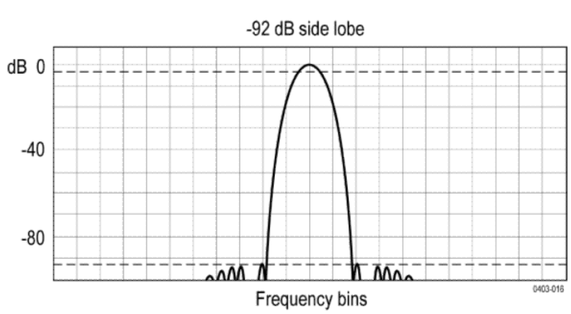
Blackman-Harris FFT window concepts

FFT windows have various resolution bandwidths and scallop losses (see the figure below). Choose the one that best allows you to view the signal characteristics you are interested in.

The Blackman-Harris window has a low amount of energy leakage compared to the other windows. Its best use is for single frequency signals to look for higher order harmonics.

Blackman-Harris window

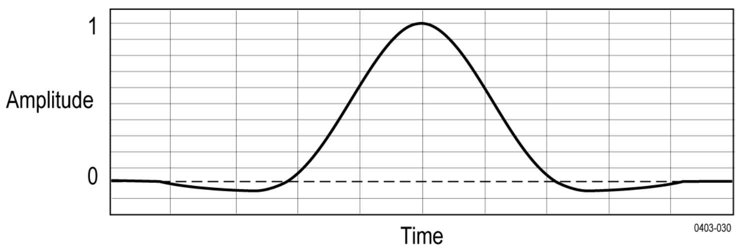


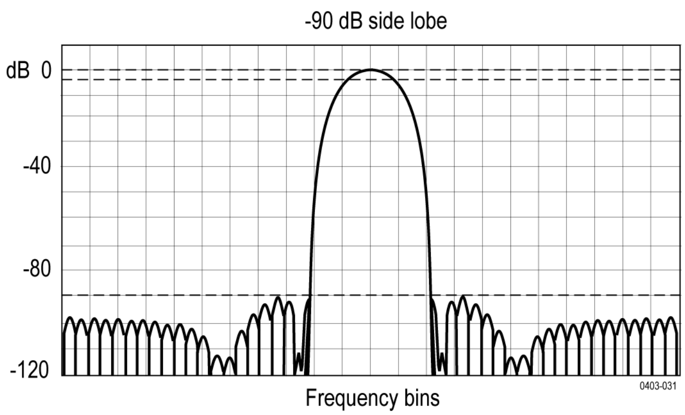


Flattop2 window

This window has the lowest scallop loss of any of the windows. It also has a wider resolution bandwidth but lower side lobe attenuation. Also, it is unique because the time domain shape has negative values.

NOTE. *The Flattop2 window is useful for high accuracy magnitude measurements for signals that do not require very narrow bandwidth.*

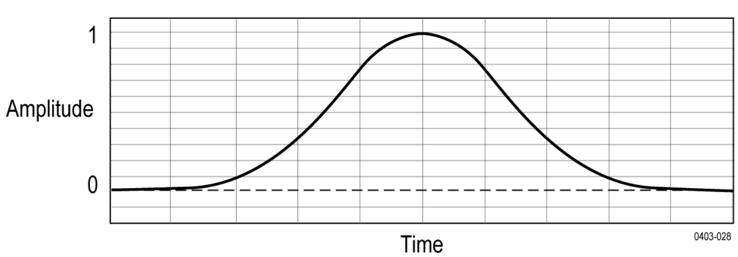


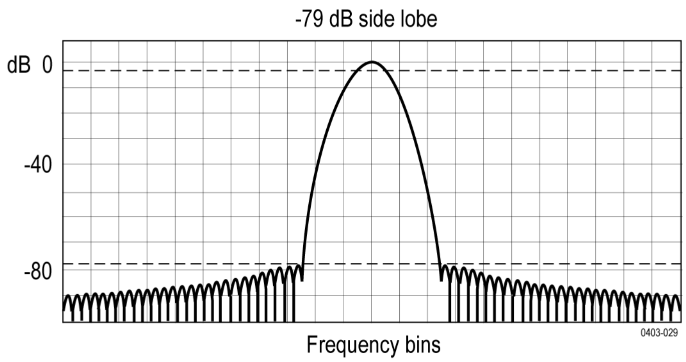


Gaussian window

This is the default window function (see the next figure). It is unique in that the time-domain shape of an exponential Gaussian function transforms into a Gaussian exponential shape in the frequency domain.

NOTE. *This window provides optimal localization in both the time and the frequency domain.*



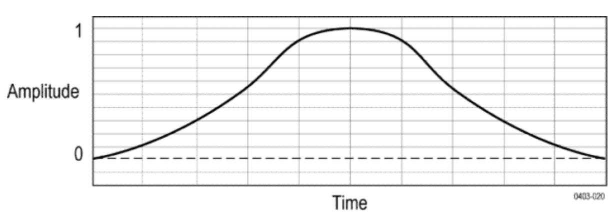


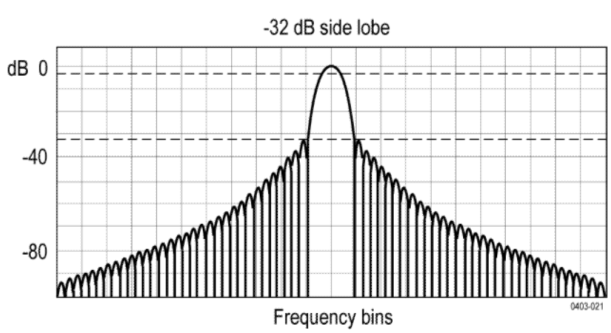
Hanning FFT window

FFT windows have various resolution bandwidths and scallop losses (see the figure below). Choose the one that best allows you to view the signal characteristics you are interested in.

The Hanning window has the narrowest resolution bandwidth, but higher side lobes. Hanning has slightly poorer frequency resolution than Hamming. Hanning is best for measuring sine, periodic, and narrow-band random noise, and transients or bursts where the signal levels before and after the event are significantly different.

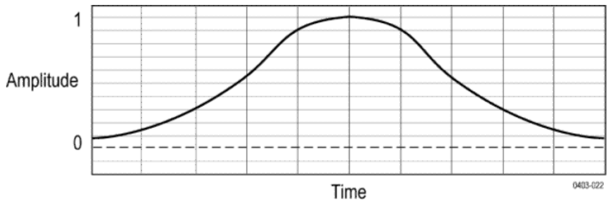
Hanning window

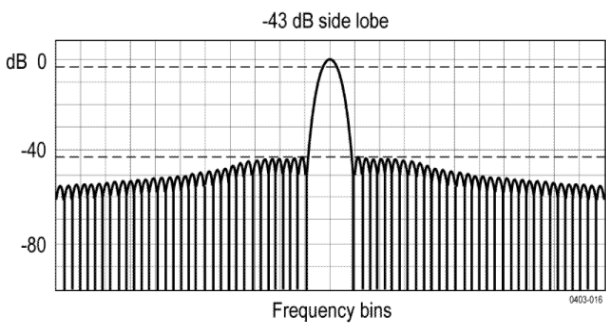




Hamming window

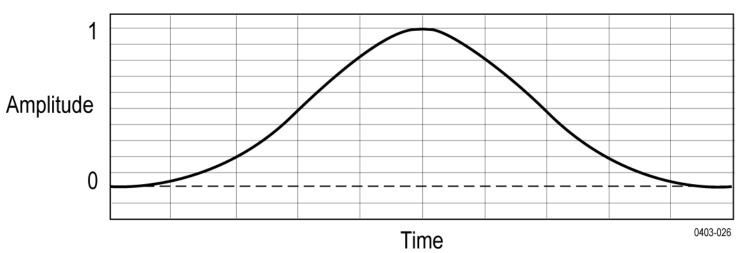
This window is unique in that the time domain shape does not taper all the way to zero at the ends. This makes it a good choice if you wanted to process the real and imaginary parts of the spectrum off line and inverse transform it back to the time domain. Because the data does not taper to zero, you can remove the effect of the window function from the result.

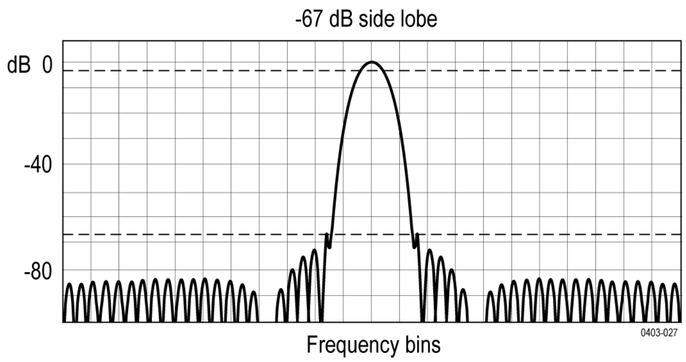




Kaiser-Bessel FFT window

A Kaiser-Bessel window balances amplitude accuracy, side lobe distance, and side lobe height. Although similar to the Blackman-Harris window, the near side lobes in a Kaiser- Bessel window tend to be higher for the same main lobe width, while the further-out side lobes are lower. This window can reveal signals close to the noise floor, and analyze two tones with close frequencies but different amplitudes.

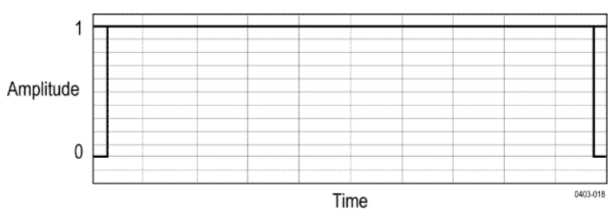


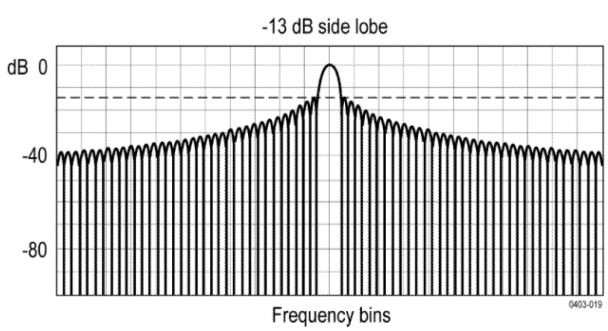


Rectangular window

This window is equal to unity (see the next figure). This means the data samples in the gate are not modified before input to the spectral analyzer. Rectangular windows are best for measuring transients or bursts where the signal levels before and after the event are nearly equal.

NOTE. *This window has the narrowest resolution bandwidth of any of the windows, but it also has the most spectral leakage and the highest side lobes.*



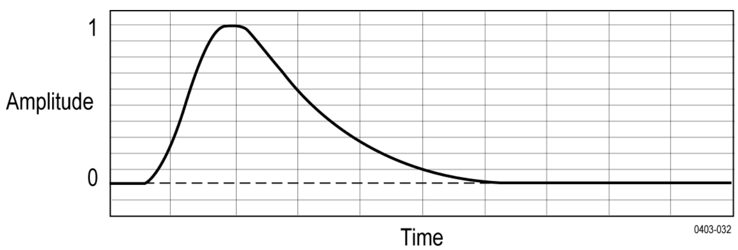


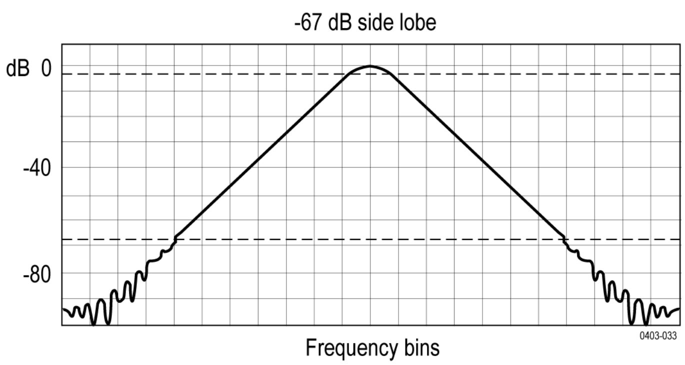
Tek-Exponential window

In the time domain, it is not a symmetrical bell shape as is the case with the other windows. Instead, it is exponential with a peak at the 20% position of the time domain gate. The frequency domain shape is triangular.

NOTE. *Use this window for impulse-response testing where the 20% position is the zero phase reference point. More of the acquired data record length is used to capture the impulse response.*

Exact details of how to compute its values were published in the article; *Impulse-Response Testing Lets a Single Test Do the Work of Thousands*, by John Pickerd, EDN magazine, April 27, 1995.





Measurement algorithms

Amplitude measurement algorithms

AC RMS measurement algorithm

AC RMS is the true Root Mean Square of the data points about the Mean (μ). This measurement can be made across the entire record or on each cycle in the record.

Area measurement algorithm

Area is the arithmetic area for one waveform. The area measured above ground is positive. The area measured below ground is negative. Remember that one waveform is not necessarily equal to one cycle. For cyclical data you may prefer to use the cycle area rather than the arithmetic area.

$$Area = \int_{Start}^{End} Waveform(t)\,dt$$

0020-012

Details of the integration algorithm are given later. *Integration algorithm* on page 369

Amplitude measurement algorithm

Amplitude is the difference between the Top value and the Base value.

Amplitude = Top - Base

Base measurement algorithm

Base is calculated using the selected Base Top method. Base is the most common data value below the midpoint of the waveform, when the default Base Top method Histogram Mode is selected. This measurement can be made across the entire record or on each cycle in the record.

Integration algorithm

The integration algorithm used by the instrument is as follows:

$$\int_{A}^{B} W(t)dt \quad \text{is approximated by} \quad \int_{A}^{B} \hat{W}(t)dt \quad \text{where:}$$

0020-025

W(t) is the sampled waveform

Ŵ(t) is the continuous function obtained by interpolation of *W(t)*

A and *B* are numbers between 0.0 and *RecordLength* – 1.0

If *A* and *B* are integers, then:

$$\int_A^B \hat{W}(t)dt = s \times \sum_{i=A}^{B-1} \frac{W(i) + W(i+1)}{2}$$

0020-026

where *s* is the sample interval.

Similarly,

$$\int_A^B (W(t))^2 dt \text{ is approximated by } \int_A^B \left(\hat{W}(t)\right)^2 dt \text{ where:}$$

0020-027

W(t) is the sampled waveform

Ŵ(t) is the continuous function obtained by interpolation of *W(t)*

A and *B* are numbers between 0.0 and *RecordLength* – 1.0

If *A* and *B* are integers, then:

$$\int_A^B \left(\hat{W}(t)\right)^2 dt = s \times \sum_{i=A}^{B-1} \frac{(W(i))^2 + W(i) \times W(i+1) + (W(i+1))^2}{3}$$

0020-028

where *s* is the sample interval.

Maximum measurement algorithm

Maximum is the maximum data point. Typically the most positive peak voltage.

Mean measurement algorithm

Mean is the arithmetic mean of the data points. Remember that one waveform is not necessarily equal to one cycle. For cyclical data you may prefer to use the cycle mean rather than the arithmetic mean.

$$Mean = \frac{\int_{Start}^{End} Waveform(t)\mathrm{dt}}{(End - Start) \times SampleInterval}$$

0020-017

Details of the integration algorithm are given in *Integration algorithm* on page 369

Minimum measurement algorithm

Minimum is the minimum data point. Typically the most negative peak voltage.

Negative Overshoot measurement algorithm

Negative Overshoot is the difference between Minimum and Base, divided by the amplitude. It is the percent that the waveform goes below base.

$$\text{Negative Overshoot} = \frac{\text{Base}-\text{Min}}{\text{Amplitude}} \text{X} 100\,\%$$

Note that overshoot values should never be negative (unless *Top* or *Base* are set out-of-range).

Positive Overshoot measurement algorithm

Positive Overshoot is the difference between Maximum and Top, divided by the amplitude. It is the percent that the waveform goes above top.

$$\text{Positive Overshoot} = \frac{\text{Max}-\text{Top}}{\text{Amplitude}} \text{X} 100\,\%$$

Note that this value should never be negative.

Peak-To-Peak measurement algorithm

Peak to peak is the difference between Maximum and Minimum.

PeaktoPeak = Max – Min

RMS measurement algorithm

RMS is the true root mean square of the data points.

$$\text{RMS} = \sqrt{\frac{(\text{data1}^2 + \text{data2}^2 \ldots)}{(\text{end - start})}}$$

Top measurement algorithm

Top is calculated using the selected Base Top method. Top is the most common data value above the midpoint of the waveform, when the default Base Top method Histogram Mode is selected. This measurement can be made across the entire record or on each cycle in the record.

Timing measurement algorithms

Burst Width measurement algorithm

Burst Width is the duration of a series of adjacent crossings of the mid reference level. The duration of a burst. Bursts are separated by a user-defined idle time.

Data Rate measurement algorithm

Data Rate is the reciprocal of Unit Interval. This measurement is made on each bit in the record.

Delay measurement algorithm

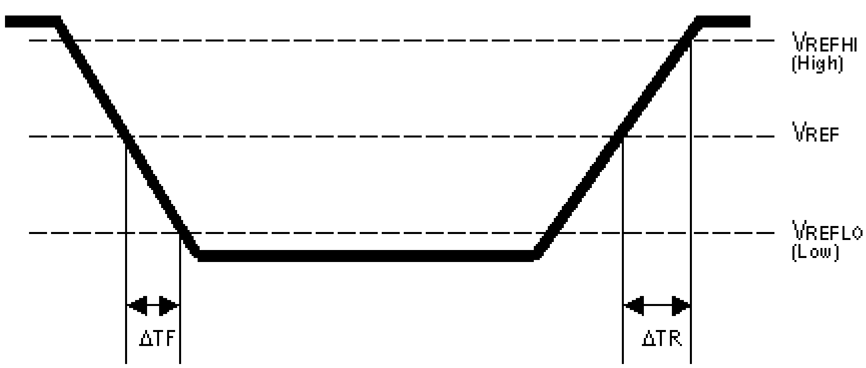
Delay is the time between a mid reference level edge on one source to a mid reference level edge on a second source. The direction of each edge can be configured by the user.

Falling slew rate

Falling Slew Rate is the rate of change in value as an edge transitions from the high or mid reference level to the mid or low reference level. The levels are configurable.

In the diagram below, the Falling Slew Rate from mid ref to low ref is calculated using the following equation:

Falling Slew Rate = (VREF - VREFLO)/ΔTF



Fall Time measurement algorithm

Fall Time is the time required for an edge to fall from the high reference level to the low reference level. By default the measurement is from reference level 90% amplitude to 10% amplitude.

The following figure shows a falling edge with the two edges necessary to calculate a Fall measurement. The figure shows the default high reference level which is 90% of Top and the default low reference level which is 10% of Base.

1. Searching from *Start* to *End*, find the first sample in the measurement zone greater than *HighRef*.
2. From this sample, continue the search to find the first (negative) crossing of edge of *HighRef*. The time of this edge is *THF*. (Use interpolation if necessary.)

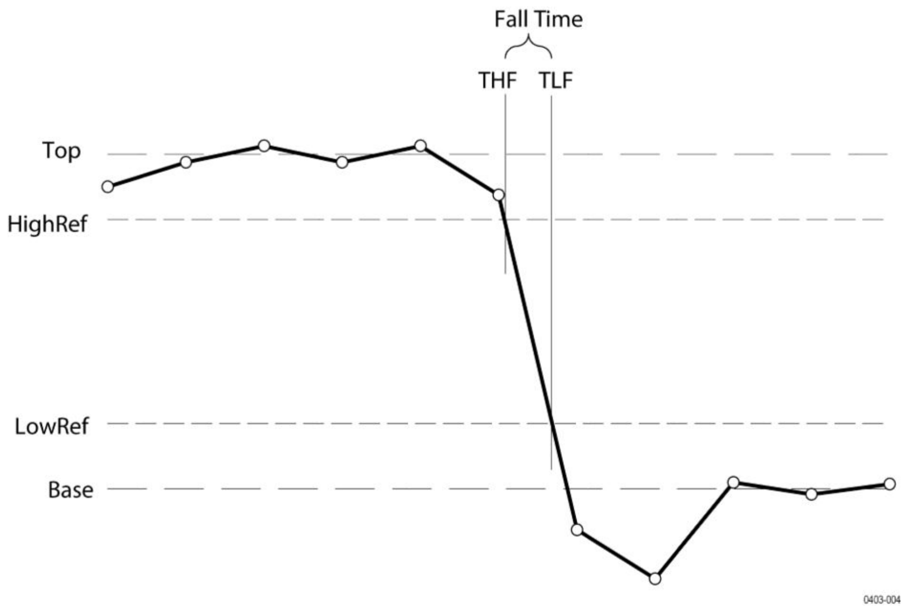


Figure 5: Fall Time

3. From *THF*, continue the search, looking for a crossing of *LowRef*. Update *THF* if subsequent *HighRef* crossings are found. When a *LowRef* crossing is found, it becomes *TLF*. (Use interpolation if necessary.)
4. *FallTime* = *TLF* – *THF*

Frequency measurement algorithm

Frequency is the reciprocal of the period. Frequency is typically measured in Hertz (Hz) where 1 Hz = 1 cycle per second.

Frequency = *1 / Period*

High Time measurement algorithm

High Time is the amount of time that a waveform cycle is above the High reference voltage level.

The application calculates the measurement using the following equation:

$$T_n^{High} = T_n^{Hi-} - T_n^{Hi+}$$

Where:

T High is the high time.

T Hi- is the High reference crossing on the falling edge.

T Hi+ is the High reference crossing on the rising edge.

Hold Time measurement algorithm

Hold Time is the time between the mid reference level crossing of the clock source (Source1) and the next mid reference level crossing of the data source (Source2). The crossings (edges) may be configured to be rising, falling or either.

The application calculates this measurement using the following equation:

$$T_n^{Hold} = T_n^{2nd} - T_i^{Main}$$

Where:

THold is the hold time.

TMain is the source 1 (clock) Mid reference edge time in the configured direction.

T2nd is the source 2 (data) Mid2 reference edge time in the configured direction.

Low Time measurement algorithm

Low Time is the amount of time that a waveform cycle is below the Low reference voltage level.

The application calculates this measurement using the following equation:

$$T_n^{Low} = T_n^{Lo+} - T_n^{Lo-}$$

Where:

T Low is the low time.

T Lo+ is the Low reference crossing on the rising edge.

T Lo- is the Low reference crossing on the falling edge.

N-Periods Duration measurement algorithm

Duration N-Periods is the time required to complete N cycles.

The source can be configured to be treated as either a clock or data waveform. Given a voltage waveform, the N-Period is calculated as follows:

If the Signal Type is Clock. The N–Period measurement calculates the elapsed time for N consecutive crossings of the mid reference voltage level in the direction specified.

The application calculates this measurement using the following equation:

$$NP_n^{Clock} = T_{n+N}^{Clock} - T_n^{Clock}$$

Where:

NPClock is the accumulated period for N clock cycles.

TClock is the VRefMid crossing time for the configured edge direction.

If the Signal Type is Data. The N–Period measurement calculates the elapsed time for N consecutive unit intervals.

The application calculates this measurement using the following equation:

$$NP_{n}^{Data} = T_{n+N}^{Data} - T_{n}^{Data}$$

Where:

NPData is the duration for N unit intervals.

TData is the VRefMid crossing time in either direction.

If *Tn+NData* does not exist for a given n, no measurement is recorded for that position.

Negative Duty Cycle measurement algorithm

Negative Duty Cycle is the ratio of the negative pulse width to the signal period expressed as a percentage.

NegativeWidth is defined in *Negative Pulse Width*, below.

If *Period* = 0 or undefined then return an error.

$$NegativeDutyCycle = \frac{NegativeWidth}{Period} \times 100\%$$

Negative Pulse Width measurement algorithm

Negative Pulse Width is the time (or distance) the signal remains below the mid reference level. It is the distance from a falling edge to the next rising edge.

Period measurement algorithm

Negative Pulse Width is the time (or distance) the signal remains below the mid reference level. It is the distance from a falling edge to the next rising edge. Period is measured in horizontal units, typically seconds.

Period = Edge3 - Edge1

Phase measurement algorithm

Phase is the ratio of the Skew between two sources to the Period of the first source. It is the amount of phase shift between edges of the two waveforms. The phase shift is expressed in degrees of the Source1 waveform cycle, where 360 degrees is one complete cycle (Period) of Source1. For best results, Source1 and Source2 should be of the same frequency or one waveform should be a harmonic of the other.

Phase is determined in the following manner for each cycle of the record:

1. The first two adjacent edges of the configured 'from' edge type are found in Source1.
2. The period of Source1 is calculated (see *Period* above). If the "from" edge is set to either, the half-period of Source1 is calculated.
3. The first edge in Source2 in the configured 'to' edge direction is found.
4. The Skew from the first Source1 edge to the next Source2 edge is calculated.

(Skew/Period)*360

5. When "from" edge is either, the calculation is (Skew/half-Period)*180.

Positive Duty Cycle measurement algorithm

Positive Duty Cycle is the ratio of the positive pulse width to the signal period, expressed as a percentage.

PositiveWidth is defined in *Positive Pulse Width*, following.

$$PositiveDutyCycle = \frac{PositiveWidth}{Period} \times 100\%$$

Positive Pulse Width measurement algorithm

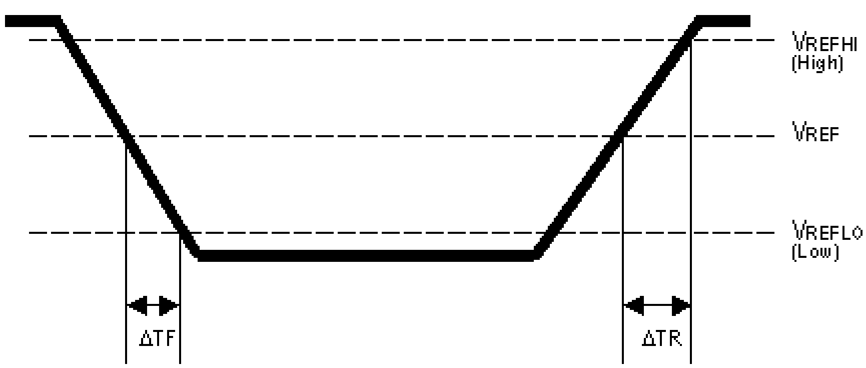
Positive Pulse Width is the time the signal remains above the mid reference level. It is the distance from a rising edge to the next falling edge.

Rising Slew Rate measurement algorithm

Rising Slew Rate is the rate of change in value as an edge transitions from the low or mid reference level to the mid or high reference level. The levels are configurable.

In the diagram below, the Rising Slew Rate from mid ref to high ref is calculated using the following equation:

(VREFHI - VREF)/ΔTR

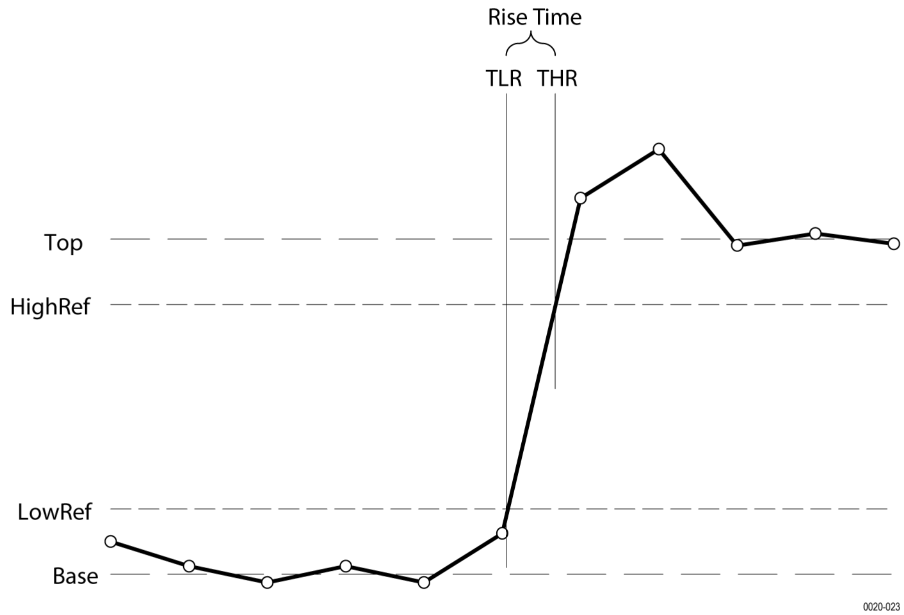


Rise Time measurement algorithm

Rise Time is the time required for an edge to rise from the low reference level to the high reference level. By default the measurement is from reference level 10% amplitude to 90% amplitude.

The following figure shows a rising edge with the two crossings necessary to calculate a Rise Time measurement.

1. Searching from *Start* to *End*, find the first sample in the measurement zone less than *LowRef*.
2. From this sample, continue the search to find the first (positive) crossing of *LowRef*. The time of this crossing is the low rise time or *TLR*. (Use linear or sin interpolation if necessary.)
3. From *TLR*, continue the search, looking for a crossing of *HighRef*. Update *TLR* if subsequent *LowRef* crossings are found. If a *HighRef* crossing is found, it becomes the high rise time or *THR*. (Use linear or sin interpolation if necessary.)
4. *RiseTime = THR – TLR*



Setup

Setup Time is the time between the mid reference level crossing of the clock source (Source1) and the closest previous mid reference level crossing of the data source (Source2). The crossings (edges) may be configured to be rising, falling or either.

The application calculates this measurement using the following equation:

$$T_n^{Setup} = T_i^{Main} - T_n^{2nd}$$

Where:

T Setup is the setup time.

T Main is the Main input (clock) Mid reference crossing time in the specified direction.

T 2nd is the 2nd input (data) Mid2 reference crossing time in the specified direction.

Skew

Skew is the time between the mid reference level crossing on Source1 to the mid reference level crossing on Source2. The direction of the edge crossing is configurable.

The application calculates this measurement using the following equation:

$$T_n^{Skew} = T_n^{Main} - T_n^{2nd}$$

Where:

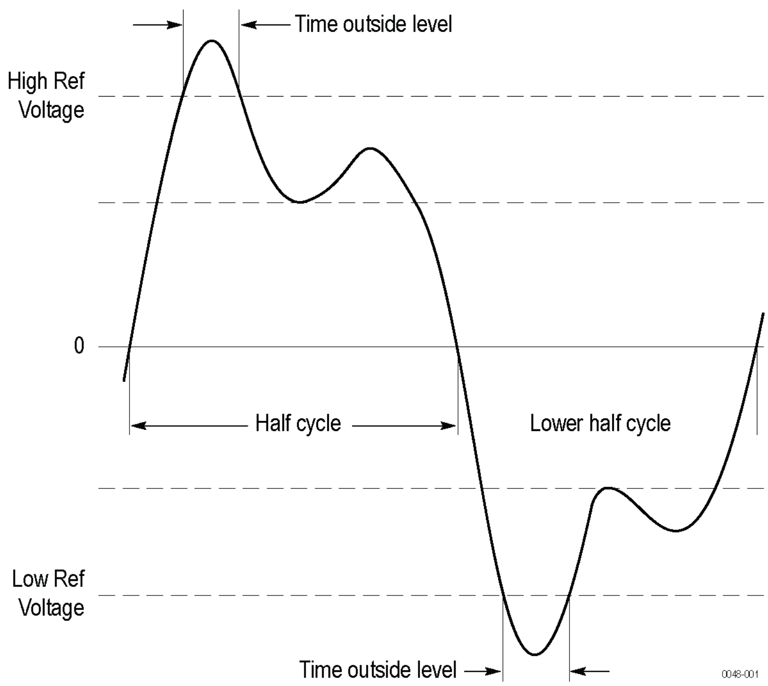
T Skew is the timing skew.

T Main is the Main input Mid reference crossing time in the configured direction.

T 2nd is the 2nd input Mid2 reference crossing time in the configured direction.

Time Outside Level measurement algorithm

Time Outside Level is the time the signal remains above the high reference level and/or below the low reference level.



Unit Interval measurement algorithm

Unit Interval is the time difference between two successive bits. This measurement is made on each bit in the record. The bits are calculated using clock recovery method constant mean.

Jitter measurement algorithms

AC Common Mode

AC Common Mode (Pk-Pk) is the peak-to-peak amplitude of the common mode of the sources. Voltage sources are typically filtered to include only the frequency components above the cutoff frequency (30 kHz). The filter can be disabled in the measurement configuration. The measurement can be configured to take one or two sources, and the cutoff frequency may be enabled or disabled. The application calculates this measurement using the following equation:

Peak-to-Peak(High Pass Filter((Source1 + Source2)/2))

Bit Amplitude measurement algorithm

Bit Amplitude is the difference between the levels of the "1" and "0" bits surrounding each transition, measured over a specified range at the center of the recovered unit interval. This measurement is made on each transition bit (Mean) or across the entire record (Mode).

Bit High measurement algorithm

Bit High is the amplitude of a "1" bit. The amplitude is measured over a user-specified portion at the center of the recovered unit interval. This measurement is made on each high bit in the record (Mean) or across the entire record (Mode).

Bit Low measurement algorithm

Bit Low is the amplitude of a "0" bit. The amplitude is measured over a user-specified portion at the center of the recovered unit interval. This measurement is made on each low bit in the record (Mean) or across the entire record (Mode).

DC Common Mode measurement algorithm

DC Common Mode is the arithmetic mean of the common mode of two sources. This measurement is made across the entire record.

Differential Crossover measurement algorithm

Differential Crossover is the voltage level of a differential signal pair at the crossover point(s). This measurement is made at each crossover point in the record.

SSC Freq Dev measurement algorithm

SSC Freq Dev is the spread spectrum clock frequency deviation from the nominal frequency in ppm (parts per million).

Clock recovery is used on the measurement. The method is constant clock mean.

SSC Modulation Rate measurement algorithm

SSC Modulating Rate is the modulating frequency of a spread spectrum clock. It is the rate that the clock frequency changes.

Clock recovery is used on the measurement. The method is constant clock mean.

TIE

TIE (Time Interval Error) is the difference in time between an edge in the source waveform and the corresponding edge in a reference clock. The reference clock is usually determined by a clock recovery process performed on the source waveform. For Explicit-Clock clock recovery, the process is performed on an explicitly identified source.

If the Signal Type is Clock. The application calculates Clock TIE measurement using the following equation:

$$TIE_n^{Clock} = T_n^{Clock} - T'{}_n^{Clock}$$

Where:

TIEClock is the clock time interval error.

TClock is the Mid reference crossing time for the specified clock edge.

T'Clock is the corresponding edge time for the specified reference clock.

If the Signal Type is Data. The application calculates Data TIE measurement using the following equation:

$$TIE_k^{Data} = T_k^{Data} - T'^{Data}_k$$

Where:

TIEData is the data time interval error.

TData is the Mid reference crossing time in either direction.

T'Data is the corresponding edge time for the specified reference clock.

The subscript k is used to indicate that there is one measurement per actual edge.

T/nT Ratio measurement algorithm

T/nT Ratio is the ratio in dB of the midpoint level of the 1st bit after each transition to the midpoints of the 2nd and subsequent non-transition bit levels prior to the next transition. This measurement is made for each non-transition bit in the record.

DCD

Duty Cycle Distortion (DCD) is the peak-to-peak amplitude for that portion of the deterministic jitter directly correlated with signal polarity, that is the difference between the mean positive edge displacement versus that on negative edges. A single DCD value is determined for each acquisition, by means of RJ-DJ separation analysis.

DDJ

Data-Dependent Jitter (DDJ) is the peak-to-peak amplitude for that portion of the deterministic jitter directly correlated with the data pattern in the waveform. A single DDJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

DJ

Deterministic Jitter (DJ) is the peak-to-peak amplitude for all timing errors that follow deterministic behavior. A single DJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

Dual Dirac deterministic jitter

Dual Dirac Deterministic Jitter (DJ–δδ) the peak-to-peak magnitude for all timing errors exhibiting deterministic behavior, calculated based on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal magnitude dirac functions (impulses). A single DJ–δδ value is determined for each acquisition, by means of RJ-DJ separation analysis.

F/2 measurement algorithm

F/2 is the peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by 2. This measurement is made across the entire record.

F/4 measurement algorithm

F/4 is the peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by 4. This measurement is made across the entire record.

F/8 measurement algorithm

F/8 is the peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by 8. This measurement is made across the entire record.

J2

J2 is Total Jitter at a Bit Error Rate (BER) value of 2.5E-3. This statistical value predicts a peak-to-peak jitter that will only be exceeded with a probability equal to the BER.

J9

J9 is Total Jitter at a Bit Error Rate (BER) value of 2.5E-10. This statistical value predicts a peak-to-peak jitter that will only be exceeded with a probability equal to the BER.

Jitter Summary measurement

The Jitter Summary measurement is a predefined set of jitter measurements displayed in a single badge. The measurements include TIE, TJ@BER, Eye Width@BER, RJ-66, DJ-66, PJ, DDJ, and DCD. This measurement also adds Eye Diagram, TIE Histogram, Tie Spectrum, and Bathtub plots to the screen.

NPJ

Non-Periodic Jitter (NPJ) is the dual-dirac magnitude of that portion of Bounded Uncorrelated Jitter (BUJ) that is not periodic. Since it is not periodic and is not correlated with the data pattern, NPJ is frequently difficult to distinguish from (Gaussian) RJ.

This component of jitter is not analyzed by default, but you can enable it by switching the jitter analysis mode to Spectral + BUJ. Since it typically requires high populations to distinguish, you may need to acquire multiple waveforms before jitter results are available when Spectral + BUJ mode is enabled.

Phase noise

The Phase Noise measurement performs a jitter measurement, converts the result into the frequency domain, and reports the rms jitter integrated between two specific frequencies selected by the user.

The phase noise measurement is defined only for clock signals. If the source waveform appears to be a data signal, a warning message will be produced but the measurement will proceed.

A Phase Noise measurement is required in order to enable the Phase Noise plot.

PJ

Periodic Jitter (PJ) is the peak-to-peak amplitude for that portion of the deterministic jitter which is periodic, but for which the period is not correlated with any data pattern in the waveform. A single PJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

RJ

Random Jitter (RJ) is the rms magnitude of all timing errors not exhibiting deterministic behavior. A single RJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

RJ 66 measurement algorithm

RJDIRAC (dual-dirac random jitter) is random jitter based on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal-magnitude Dirac functions. This measurement is made across the entire record.

SRJ

Sub-Rate Jitter is periodic jitter at a rate that integrally divides the data rate. For example, if the data rate is F bits/second, sub-rate jitter components could occur at F/2 or F/4. It typically occurs when a serial data stream is formed by multiplexing (interleaving) an integral number of lower-rate bit streams together, although there can be other causes. Sub-rate jitter is a sub-component of PJ.

The SRJ measurement is the peak-to-peak amplitude for the sum of all F/N jitter components that are tracked by DPOJET. Since different F/N components are correlated with each other, the peak-to-peak SRJ depends on relative phases and is not simply the sum of the individual F/N components.

The SRJ measurement always tracks and accounts for N = 2, 4 and 8 regardless of whether the corresponding F/N measurements have been selected.

TJ@BER

Total Jitter at a specified Bit Error Rate (BER). This extrapolated value predicts a peak-to-peak jitter that will only be exceeded with a probability equal to the BER. It is generally not equal to the total jitter actually observed in any given acquisition. A single TJ@BER value is determined for each acquisition, by means of RJ-DJ separation analysis.

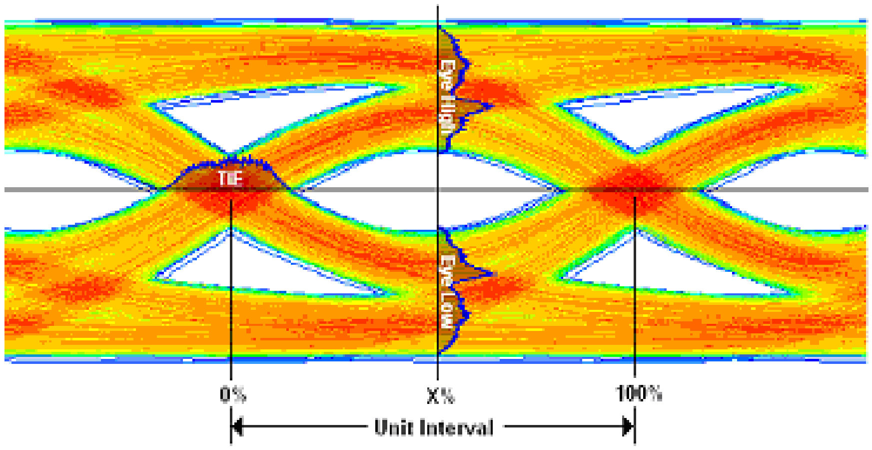
Eye measurement algorithms

Eye Height measurement algorithm

Eye Height is the minimum vertical eye opening at the mid of the unit interval. This measurement is made across the entire record.

Eye high

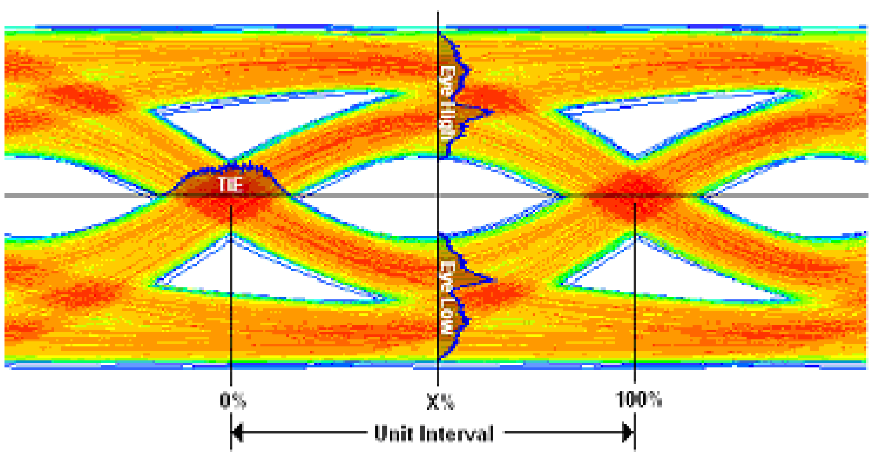
Eye High calculates the voltage at a selected horizontal position across the unit interval, for all High bits in the waveform. You specify the offset at which the measurement takes place from 0% to 100% of the unit interval. Configure the measurement to include all bits, only transition bits, or only non-transition bits. (Note that some of the waveform can be omitted from the measurement due to initialization of clock recovery or filtering.) A histogram of the Eye High measurement corresponds to a vertical slice through the upper half of a three-dimensional eye diagram.



NOTE. *This illustration shows how the measurement is made, and does not represent how the oscilloscope actually displays an eye diagram or histograms on an eye diagram plot.*

Eye low

Eye Low calculates the voltage at the selected horizontal position across the unit interval, for all Low bits in the waveform. A histogram of the Eye Low measurement corresponds to a vertical slice through the lower half of a three-dimensional eye diagram.



NOTE. *This illustration shows how the measurement is made, and does not represent how the oscilloscope actually displays an eye diagram or histograms on an eye diagram plot.*

Eye Width measurement algorithm

Eye Width is the minimum horizontal eye opening at the user-specified reference level. This measurement is made across the entire record.

Height@BER

Height@BER is the Eye Height at a specified Bit Error Rate (BER). This extrapolated value predicts a vertical eye opening that will be violated with a probability equal to the BER. It is generally not equal to the eye height actually observed in any given acquisition. A single Height@BER value, in the given interval, is determined for each acquisition by means of Q-scale extrapolation.

Q-factor

Quality Factor is the ratio of eye size to noise.

The final measurement value would be computed according to the equation below:

```
Q-factor = [mean(EyeHigh) - mean(EyeLow)] / [stddev(EyeHigh) + stddev(EyeLow)]
```

Where:

Eye High: the sample values of positive UI at x%.

Eye Low: the sample values of negative UI at x%.

For more details refer Eye Height

Width@BER

Width@BER is the Eye Width at a specified Bit Error Rate (BER). This extrapolated value predicts a horizontal eye opening that will be violated with a probability equal to the BER. It is generally not equal to the eye width actually observed in any given acquisition. A single Width@BER value is determined for each acquisition, by means of RJ-DJ separation analysis.

Power measurement algorithms

Input analysis algorithms

Power Quality measurement algorithm. The Power Quality measurement calculates the Frequency and RMS values of the voltage and current, Crest Factors of the voltage and current, True Power (TrPwr), Reactive Power (RePwr), Apparent Power (ApPwr), Power Factor (PF), and Phase Angle (θ) of the AC signal.

- **RMS Voltage**: The application calculates the RMS voltage using the following equation:

$$V_{RMS} = \sqrt{\frac{1}{N}\sum_{n-0}^{N-1} v^2(n)}$$

Where:

V_{RMS} is the RMS voltage in Volts.

N is the number of samples.

n is the data point.

v(n) is the absolute value of the voltage at the particular data point.

NOTE. *The voltage RMS is for all the time domain cycles in the acquisition.*

- **RMS Current**: The application calculates the RMS current using the following equation:

$$V_{RMS} = \sqrt{\frac{1}{N}\sum_{n-0}^{N-1} v^2(n)}$$

Where:

I_{RMs} *is the RMS current in Amps*.

N is the number of samples.

n is the data point.

i(n) is the absolute value of the current at the particular data point.

- **Apparent Power (ApPwr)**: It is the product of the RMS voltage and current (mathematically, the absolute value of the vector sum of the true and reactive power), measured in Volt-Amperes or VA. The application calculates the **Apparent Power** (ApPwr) using the following equation:

$$\text{ApPwr} = V_{RMS} \times I_{RMS}$$

Where:

ApPwr is the Apparent Power, Volt-Amperes (VA).

V_{RMS} *is the root mean square of the voltage*.

I_{RMS} *is the root mean square of the current*.

- **Reactive Power (RePwr)**: The reactive power or the imaginary power delivered to and temporarily stored in the reactive (inductive or capacitive) elements of the load, measured in units of Volt-Amperes-Reactive or VAR. The application calculates the Reactive power using the following equation:

$$RePwr = \sqrt{ApPwr^2 - TpPwr^2}$$

Where, *RePwr* is the Reactive Power, Volt-Amperes-Reactive or VAR.

- **Power Factor (PF)**: It is calculated using the following equation:

$$PF = \frac{TrPwr}{ApPwr}$$

Where, *PF* is the Power Factor.

- **Crest Factor (CF)**: It is the ratio of the peak voltage value of the signal to the RMS value of the signal. Use the following equation to calculate the crest factor for the voltage and current:

$$VCF = \frac{V_{pk}}{V_{RMS}}$$

Where:

VCF is the Voltage Crest Factor.

V_{PK} *is the peak value of the voltage*.

V_{RMS} *is the Root Mean Square of the voltage.*

$$ICF = \frac{I_{pk}}{I_{RMS}}$$

Where:

ICF is the Current Crest Factor.

I_{RMS} *is the Root Mean Square of the current*.

- **Phase Angle (σ)**: It is the angle (-90 to +90) whose cosine is the true power factor. Unit of Phase Angle is degrees. The angle is positive if the Ch1 waveform (typically voltage) leads the Ch2 waveform (typically current). The angle is negative if the Ch1 waveform lags behind the Ch2 waveform. The application calculates the phase angle using the following equation:

$$\emptyset = Tan^{-1}[\frac{RePwr}{ApPwr}]$$

Where, *σ* is the Phase Angle, Degree.

Harmonics algorithm. Harmonics are the sinusoidal voltages or currents having frequencies that are integer multiples of the frequency at which the supply system is designed to operate (termed the fundamental frequency). Distorted waveforms can be decomposed into sum of the fundamental frequency and its harmonics.

- The measurement uses the Discrete Fourier Transform (DFT) to calculate the Real component (Re(k)) and Imaginary component Im(k).The Real component (Re(k)) and Imaginary component Im(k) are calculated using the following equation:

$$Re[k] = \sum_{i=0}^{N-1} x[i]\cos(2\pi ki/N)$$

$$Im[k] = -\sum_{i=0}^{N-1} x[i]\sin(2\pi ki/N)$$

  Where:

  i is the Index of the input data value.

  k is the Index of the harmonics, k index is calculated using the harmonics number.

  x[i] is the discreet set of acquire time samples, Re[k] is the Real component of kth harmonic.

  Im[k] is the Imaginary component of kth harmonic.

- **Harmonics Fk** is calculated using the following equation:

$$F[k] = \sqrt{2\left\{\left(\frac{Re(k)}{N}\right)^2 + \left(\frac{Im(k)}{N}\right)^2\right\}}$$

  Where:

  F[k] is the kth harmonic, in Amp/Volt.

  Unit of F[k] for voltage signal is Volt and for current signal is Ampere.

  Harmonics is converted into dB using the following equation:

$$F[k]^{dB} = 20 \times log_{10}(F[k]) + 120dB$$

  $F[k]^{dB}$ is the k^{th} harmonic , dBA or dBV based on the harmonics input source configuration.

  NOTE. *In case of 'AM 14' standard, acquired signal is divided into 15 chunks and all harmonics are calculated for each chunk. For each individual harmonics maximum is taken out of 15 values.*

- **RMS**: The RMS of harmonics is calculated in unit of volt or ampere using the following equation:

$$RMS = \sqrt{F1^2 + F2^2 + \cdots .. + FOH^2}$$

  F1 is the fundamental harmonics or line harmonics.

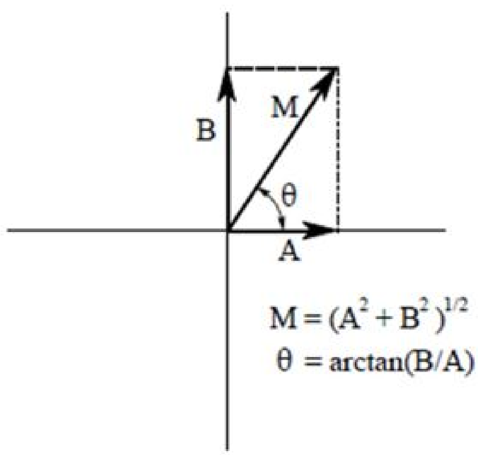
- **Total Harmonic distortion (THD-F)**: It is measured as the ratio to the RMS value of the fundamental component of the source waveform. Reported as a percentage and calculated using the following equation:

$$THD - F = \sqrt{\frac{RMS^2 - F^2[k]}{F1}} \times 100\%$$

- **Total Harmonic Distortion (THD-R)**: It is measured as a ratio to the RMS value of the source waveform. Reported as a percentage and calculated using the following equation:

$$THD - R = \sqrt{\frac{RMS^2 - F^2[k]}{RMS}} \times 100\%$$

- **Phase calculation**: The frequency domain can be expressed in polar form. In this notation, real (Re[ ])& imaginary (Im[ ]) component in frequency domain are replaced with two other arrays, called the Magnitude is Mag[i], and the Phase is written as: σ. The magnitude and phase are a pair-for-pair replacement for the real and imaginary parts.



$$Mag[i] = \sqrt{Rel[i]^2 + Img[i]^2}$$

Where:

i is the Index of harmonics.

Rel[i] is the Real component of the harmonics (Cos frequency).

Img[i] is the Imaginary component of the harmonics (Sin frequency).

$$Phase[k] = tan\frac{Img[k]}{Rel[k]}$$

If, Rel [k] < 0 & Im[k] < 0 Phase[k] = Phase[k] - π

If, Rel[k] < 0 & Im[k] > 0 Phase[k] = Phase[k] + π

- **Partial Odd Harmonics Current (POHC(M))**:

  For the 21st standard higher odd order harmonics, the average values obtained for each individual odd harmonic over the full observation period, are calculated from the acquired waveform. The measured partial odd harmonic current does not exceed the partial odd harmonic current which can be calculated from the applicable limits.

$$\mathrm{POHC(M)} = \sqrt{\sum_{k=21,23}^{OH} F[k]^2}$$

  Where:

  M is the measured value.

  POHC(S) and POHC(L) is the (S) is pass and fail status based on (L) limits.

Amplitude analysis measurement algorithms

Cycle Base measurement algorithm. Cycle Base is calculated using the selected Base Top method. Cycle Base is the most common data value below the midpoint of the waveform, when the default Base Top method Histogram Mode is selected. This measurement is made across each cycle in the record.

Cycle Maximum measurement algorithm. Cycle Maximum is the maximum data point. Typically the most positive peak voltage.

Measurements are calculated on each cycle within the record.

Cycle Minimum measurement algorithm. Cycle Minimum is the minimum data point. Typically the most negative peak voltage.

Measurements are calculated on each cycle within the record.

Cycle Pea-to-Peak measurement algorithm. Cycle Peak to peak is the difference between Cycle Maximum and Cycle Minimum calculated for each cycle.

PeaktoPeak = Max – Min

Measurements are calculated on each cycle within the record.

Cycle Top measurement algorithm. Cycle Top is calculated using the selected Base Top method. Cycle Top is the most common data value above the midpoint of the waveform, when the default Base Top method Histogram Mode is selected. This measurement is made across each cycle in the record.

Cycle Amplitude measurement algorithm. Cycle Amplitude is the difference between the Top value and the Base value. This is applicable for each cycle.

Cycle Amplitude = Top - Base

Timing analysis measurement algorithms

Frequency measurement algorithm. Frequency is the reciprocal of the period. Frequency is typically measured in Hertz (Hz) where 1 Hz = 1 cycle per second.

Frequency = 1 / Period

Negative Duty Cycle measurement algorithm. Negative Duty Cycle is the ratio of the negative pulse width to the signal period, expressed as a percentage.

$$NegativeDutyCycle = \frac{NegativeWidth}{Period} \times 100\%$$

NegativeWidth is defined in *Negative Pulse Width measurement algorithm*.

If *Period* = 0 or undefined then return an error.

Negative Pulse Width measurement algorithm. Negative Pulse Width is the time the signal remains below the mid reference level. This measurement is made on each cycle in the record.

Period measurement algorithm. Period is the time required to complete a cycle. This measurement is made on each cycle in the record.

Positive Duty Cycle measurement algorithm. Positive Duty Cycle is the ratio of the positive pulse width to the signal period, expressed as a percentage.

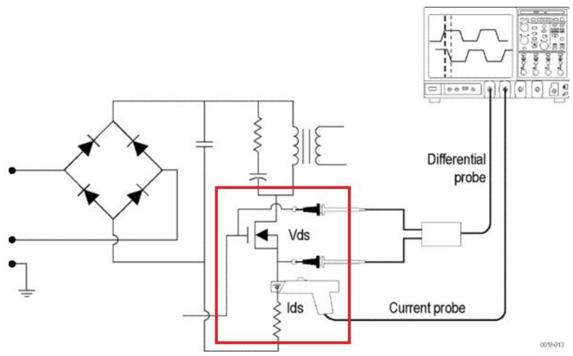
$$PositiveDutyCycle = \frac{PositiveWidth}{Period} \times 100\%$$

PositiveWidth is defined in *Positive Pulse Width algorithm*.

Positive Pulse Width measurement algorithm. Positive Pulse Width is the time the signal remains above the mid reference level. It is the distance from a rising edge to the next falling edge.

Switching analysis algorithms

Switching Loss algorithm. Switch-Mode Power Supply (SMPS) design has three types of losses, they are Turn-On (T_{on}), Turn-off (T_{off}), and Conduction loss (Cond). To achieve the maximum efficiency, losses should be reduced. This section details about the basics of Switching Loss Analysis. A simplified SMPS schematic is shown below:



SMPS circuit diagram shows the points where switching loss can be measured. After full wave rectification, the current signal should pass through the harmonic standard and enters for DC conversation. MOSFIT plays an important role to meet the design of SMPS.

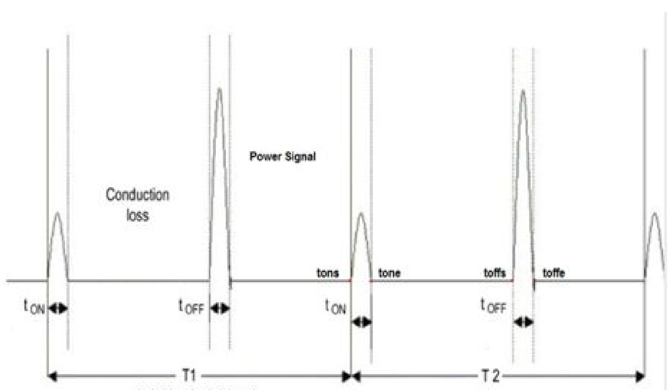
Regions of T_{on}, T_{off}, and Conduction loss (Cond) with voltage source (Vds) and current source (Ids) are shown in Switching Loss T_{on}, T_{off}, and Conduction loss regions.

All the switching losses are measured on Power signal, based on Vds and Ids transition.

T_{on} Loss region: When Vds starts rolling towards zero, the Ids starts to roll upward.

T_{off} Loss region: When Vds starts rolling upwards, the Ids start to roll towards zero.

Conduction Loss region: The region when the Ids is high and Vds is low.



T1 is the First switching cycle.

T2 is the Second switching cycle.

- **T_{on} and T_{off} losses**

  T_{on} and T_{off} losses per switching cycle are computed as in the following equations:

$$T_{oni} = f_{swi} \times \int_{T_{on}-Start_i}^{T_{on}-Stop_i} (V_{ds} \times I_{ds})\, dt\ Watt$$

$$T_{offi} = f_{swi} \times \int_{T_{off}-Start_i}^{T_{off}-Stop_i} (V_{ds} \times I_{ds})\, dt\ Watt$$

Where,

T_{oni} is the Turn on loss of the i^{th} switching cycle, in watt.

T_{OFFi} is the Turn off loss of the i^{th} switching cycle, in watt.

fswi is the Switching frequency of the ith switching cycle, in Hz.

(T_{off}-$Stop_i$) is the stop point of T_{off} region of the i^{th} switching cycle, in time unit.

(T_{on}-$Stop_i$) is the stop point of T_{on} region of the i^{th} switching cycle, in time unit.

(T_{on}-$Start_i$) is the start point of a power T_{on} region of the i^{th} switching cycle.

(T_{off}-$Start_i$) is the start point of TOffi region, in time unit Vds - Voltage drain current, in Volts.

Ids is the drain current, in Amps.

- **Energy Loss Computation**

  Energy loss computation for T_{on} and T_{off} are calculated using the following equation:

$$T_{Eoni} = f_{swi} \times \int_{T_{on}-Start_i}^{T_{on}-Stop_i} (V_{ds} \times I_{ds})\, dt\ Watt$$

$$T_{Eoffi} = f_{swi} \times \int_{T_{off}-Start_i}^{T_{off}-Stop_i} (V_{ds} \times I_{ds})\, dt\ Watt$$

  Where:

  T_{Eoni} is the i^{th} switching cycle turn on energy loss in joule.

  T_{Eoffi} is the i^{th} switching cycle turn off energy loss in joule.

- **Computation of Conduction**

  Conduction is computed as RDS(on) value for the MOSFET is used to calculate total loss in the application. To measure conduction loss and energy in a MOSFET, using the following equation:

$$Cond_i = f_{swi} \times \int_{T_{on}-Stop_i}^{T_{off}-Start_i} \left(R_{ds}(on) \times I_{ds}^2\right) dt\ Watt$$

$$Cond_{Ei} = \int_{T_{on}-Stop_i}^{T_{off}-Start_i} \left(R_{ds}(on) \times I_{ds}^2\right) dt\ Joule$$

  Where:

  R_{DS}(on) is the Dynamic resistance, in Ω.

  $Cond_i$ is the Conduction Loss in Watt, $Cond_{Ei}$ is the Conduction Energy Loss in Joule.

  $Cond_i$ is the i^{th}switching cycle conduction loss in watt. $Cond_{Ei}$ is the i^{th}switching cycle conduction energy in joule.

  f_{swi} is the i^{th} switching cycle frequency, in Hz I is the Cycle number.

  Nc is the Number of conduction cycles.

To measure conduction loss in a BJT/IGBT, using the following equation:

$$Cond_i = f_{swi} \times \int_{T_{on}-Stop_i}^{T_{off}-Start_i} (V_{ce}(SAT) \times I_{ds})\, dt\ watt$$

$$Cond_{ei} = \int_{T_{on}-Stop_i}^{T_{off}-Start_i} (V_{ce}(SAT) \times I_{ds})\, dt\ joule$$

Where, V_{CE}(sat) is the voltage in volt, which should be configure in application.

- **Computation of Average Loss and Total Loss**

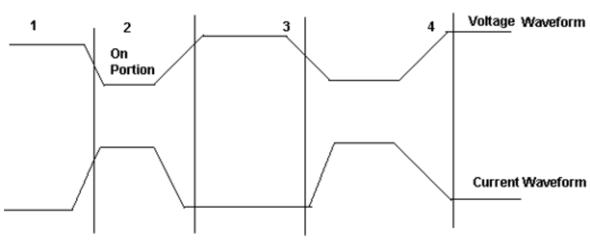
  Average and Total loss are calculated using the following equation:

$$T_{on_{Avg}} = \frac{1}{Nc} \sum_{i=1}^{Nc} T_{on_i}$$

$$T_{off_{Avg}} = \frac{1}{Nc} \sum_{i=1}^{Nc} T_{off_i}$$

$$Cond_{Avg} = \frac{1}{Nc} \sum_{i=1}^{Nc} Cond_i$$

$$TotalLoss = T_{on_{Avg}} + T_{off_{Avg}} + Cond_{Avg}$$



Total Switching Loss=Ton Loss+Toff Loss+Conduction Loss

Concept to identify Ton and Toff using gate voltage for edge analysis:

Use gate voltage for edge analysis with default 50% edge level and hysteresis 10%.

- To find the start of Ton: The start of the Ton is 5% or 1.5 V whichever is lower on the rise slope of the gate voltage.
- To find the stop of Ton: The start index on the switch voltage is 5% or 1.5 V of the rise slope gate voltage. Move forward on the switch voltage from the start index until 5% or the configured level is met.
- To find the start of Toff: The start index is 80% of the gate voltage. From the Stop index, search for 5% of the switch voltage (on rise slope).
- To find the stop of Toff: The 80% of the gate voltage is start index. From this start index on switch current (fall slope) move forward until 5% of the max (switch current) is met.

dv/dt algorithm. dv/dt represents the rate at which the voltage changes during switching. The application uses the math feature to provide a differentiation waveform of the voltage input.

When you run the measurement, the application calculates dv/dt for the first edge by taking the default levels as 10% and 90% and displays the results.

Select a specific section of the waveform on the live signal by providing inputs for high and low levels in terms of percentage and absolute value of voltage and current. Select the edge of interest by viewing it visually on the oscilloscope. You can also enter the edge number on the results panel. The application displays the results for the selected edge and levels on the results panel.

The application calculates dv/dt using the following equation:

$$\frac{dv}{dt} = \frac{v(R_t) - v(R_B)}{R_T - R_A}$$

Where:

X is the timing values.

Y is the vertical (voltage) values of the waveform data between the cursors.

di/dt algorithm. di/dt measurement represents the rate at which the current changes during switching. The application uses the oscilloscope's built in math feature to provide a differentiation waveform of the current input.

When you run the measurement, the application calculates di/dt for the first edge by considering the default levels as 10% and 90% and displays the results.

Select a specific section of the waveform on the live signal by providing inputs for high and low levels in terms of percentage and absolute value of voltage and current. Select the edge of interest by viewing it visually on the oscilloscope. You can also enter the edge number on the results panel. The application displays the results for the selected edge and levels on the results panel.

The application calculates di/dt using the following equation:

$$\frac{di}{dt} = \frac{i(R_t) - i(R_B)}{R_T - R_A}$$

Where:

I is the timing value.

t is the vertical values of the waveform data.

SOA algorithm. SOA plots the graph of the voltage and current waveform. You can configure SOA mask by creating voltage and current co-ordinates in the mask table. You can save and recall mask files. The extension of the mask file name is .pwrmsk.

SOA has two outputs:

1. SOA X-Y plot with mask showing hits on the mask.
2. SOA summary on the measurement badge displays the number of hits with Pass/Fail status.

Output analysis algorithms

Line Ripple algorithm. The Ripple measures using the configured line frequency (50, 60, 400 Hz), peak-to-peak noise value on the output DC signal. In Ripple Line, the time base is set to have three cycles of 50 Hz or 60 Hz in the input waveform. In Ripple switching, depending on the switching frequency, input sets the time base to display four cycles.

Depending upon the coupling type you select, the application sets the required offset and adjusts the vertical scale to the appropriate sensitivity. The application measures the peak-to-peak value and displays the result.

Switching Ripple algorithm. It measures the ripple at operating switching frequency. This can be configured up to 1 MHz.

References

Power badge error and warning messages (Option 5-PWR, SUP5-PWR, 5-PS2 only)

These tables provide information to help resolve error or warning messages that appear on the power measurement badges.

Error messages displayed on the power measurement badges

Error Message	Cause	Suggestion
Empty input	Oscilloscope is waiting for a Single Seq trigger.	Check that the oscilloscope has a valid input waveform.
Input source mismatch	Incorrect combination of live (active) and Ref source waveforms. In the case of two Ref waveforms, the two waveforms recalled must have the same record length and have been acquired with the same timebase setting.	Acquire two waveforms of the same type (active or ref) Reacquire the Ref waveforms with the same time base and record length.
Too few edges	No edges found on the input waveform as the waveform could be very noisy.	Increase the Hysteresis band and rerun the measurement. The Hysteresis values is set in the measurement configuration menu Reference Levels panel. Change the Acquisition mode and BW limit to reduce the noise in the waveform.
Error from Frequency measurement	Not able to measure frequency value on the input waveform due to noisy waveform.	Adjust the measurement Hysteresis band to compute edges properly so that frequency can be measured. The Hysteresis values is set in the measurement configuration menu Reference Levels panel.
Error from RMS measurement	Not able to measure RMS value on the input waveform due to noisy waveform.	Adjust the measurement Hysteresis band to compute edges properly so that RMS can be measured. The Hysteresis values is set in the measurement configuration menu Reference Levels panel.
Error from MaxElement measurement	Not able to measure MIN/MAX value on the input waveform due to noisy waveform.	Adjust Hysteresis band to compute edges properly so that MIN/MAX can be measured. The Hysteresis values is set in the measurement configuration menu Reference Levels panel.
Invert probing points	Polarity mismatch between the probe to test points on th DUT	Check and inverse the probe polarity to match with DUT test points.
Too few edges	Scope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Also make sure to use Power Autoset since it sets up the scope parameters properly.

Error Message	Cause	Suggestion
Insufficient sampling rate	Not enough sampling rate used to capture signal	Increase the sample rate proportional to input signal frequency. You can use Power Autoset to setup the scope parameters properly.
Too few cycles	Not enough sampling rate used to capture signal	Increase the sample rate proportional to input signal frequency. You can use Power Autoset to setup the scope parameters properly.
Not enough data	Scope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Also make sure to use Power Autoset since it sets up the scope parameters properly.
Input source mismatch. Current signal expected in current source.	Mismatch between input waveform and wrong source set. For example, if current waveform is used for voltage configuration, this error is shown.	Make sure a Current waveform is set as the input to Current Harmonics measurement.
Input source mismatch. Voltage signal expected in voltage source.	Mismatch between input waveform and wrong source set. For example if voltage waveform is used for current configuration, this error is shown.	Make sure a Voltage waveform is set as the input to Voltage Harmonics measurement.
Invalid input	Incorrect combination of live (active) and Ref source waveforms. In the case of two Ref waveforms, the two waveforms recalled must have the same record length and have been acquired with the same timebase setting.	Acquire two waveforms of the same type (active or ref) Reacquire the Ref waveforms with the same time base and record length.
Insufficient horizontal resolution	Not enough sampling rate used to capture signal	Increase the sample rate proportional to input signal frequency. You can use Power Autoset to setup the scope parameters properly.
Invalid signal frequency	Not able to measure frequency value on the input waveform due to noisy waveform	Adjust Hysteresis band to compute edges properly so that frequency can be measured. The Hysteresis values is set in the measurement configuration menu Reference Levels panel.
Not enough data	Scope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Also make sure to use Power Autoset since it sets up the scope parameters properly.
Warning from RMS measurement	Not able to measure RMS value on the input waveform due to noisy waveform	Adjust Hysteresis band to compute edges properly so that RMS can be measured. The Hysteresis values is set in the measurement configuration menu Reference Levels panel.
Too few cycles	Scope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Also make sure to use Power Autoset since it sets up the scope parameters properly.
Not enough data	Table results will not show values.	Check input waveforms and configuration. Recommended to run Power Autoset and then observe the measurement results.

Warning messages displayed on the Power measurement badges

Warning Message	Cause	Suggestion
No data in range	There are no data between the two cursors.	Place the cursors appropriately.
Warning from Frequency measurement	Not able to measure frequency value on the input waveform due to noisy waveform.	Adjust Hysteresis band to compute edges properly so that frequency can be measured. The Hysteresis values is set in the measurement configuration menu Reference Levels panel.
Warning from RMS measurement	Not able to measure RMS value on the input waveform due to noisy waveform.	Adjust Hysteresis band to compute edges properly so that RMS can be measured. The Hysteresis values is set in the measurement configuration menu Reference Levels panel.
Warning from MaxElement measurement	Not able to measure MIN/MAX value on the input waveform due to noisy waveform.	Adjust Hysteresis band to compute edges properly so that MIN/MAX can be measured. The Hysteresis values is set in the measurement configuration menu Reference Levels panel.
Voltage source expected	Mismatch between input waveform and measurement. For example if a current waveform is used for dv/dt, this warning is shown.	Make sure a voltage waveform is set as the voltage source input.
Current source expected	Mismatch between input waveform and measurement. For example if a voltage waveform is used for di/dt, this warning is shown.	Make sure a current waveform is set as the current source input.
Warning from RMS measurement	Not able to measure RMS value on the input waveform due to noisy waveform	Adjust the Hysteresis band to compute edges properly so that RMS can be measured. The Hysteresis values is set in the measurement configuration menu Reference Levels panel.
Invalid Mask	Mask is not of the standard format, cannot have closed mask coordinates or intersection of the coordinate points	Recreate the mask from the SOA mask configure table such that inner mask coordinates do not intersect.

Index

B

C

D

E

F

G

H

I

J

N

O

P

T

U

V

W

X

Z