



**5/6 Series MSO  
(MSO54, MSO56, MSO58, MSO58LP, MSO64)  
Printable Help**

Supports Product Firmware V1.14.x



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

TEKTRONIX and TEK are registered trademarks of Tektronix, Inc.

FlexChannel, TekVPI, FastAcq, and e\*Scope are registered trademarks of Tektronix, Inc.

FastFrame and TekSecure are trademarks of Tektronix, Inc.

## **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](http://www.tek.com) to find contacts in your area.

---

# Table of Contents

TEKTRONIX SOFTWARE LICENSE AGREEMENT .....	17
Welcome to the 5/6 Series MSO instrument help .....	xxi

## Product documents and support

Related documents .....	1
Product support and feedback .....	3

## Accessories

Standard accessories .....	5
Recommended accessories .....	6
Recommended probes .....	8

## Options

Bandwidth options .....	11
Record length options .....	14
Microsoft Windows 10 operating system option .....	16
Arbitrary Function Generator (AFG) (optional) .....	17
Enhanced instrument security (factory option) .....	18
Serial bus and trigger options .....	19
Compliance testing options .....	20
Advanced Jitter Analysis option .....	20
Advanced Power Analysis option .....	21
DDR option .....	21
Power cord options .....	22
Service options .....	23
How to install an option license .....	24
How to uninstall (return) an option license .....	26

## Install your instrument

Check shipped accessories .....	29
Safely rotate the handle .....	30
Operating requirements .....	31
Input signal requirements .....	32
Powering the oscilloscope .....	33
Check that oscilloscope passes power-on self tests .....	34

Installing and activating Windows 10 option .....	35
Install the Windows SSD drive .....	35
Powering on Windows for the first time .....	36
Activating Windows .....	37
Differences between Windows and base instrument user interfaces .....	38
Updating the Windows TekScope application software .....	38
Secure (lock) the oscilloscope .....	39
Connecting Probes .....	40
Rackmount information .....	42

## Getting acquainted with your instrument

Front panel controls and connectors .....	43
MSO58LP front panel controls and connections .....	54
Rear panel connections .....	56
The user interface screen .....	57
The user interface elements .....	59
Badges .....	63
Moving waveform and measurement badges .....	69
Group signal badges in the Settings bar .....	71
Configuration menus .....	72
The Zoom user interface elements .....	74
Using the touch screen interface for common tasks .....	75
Accessing application help .....	77

## Configure the instrument

Set the time zone and clock readout format .....	79
Functional check .....	80
Download and install the latest firmware .....	81
Run Signal Path Compensation (SPC) .....	82
Compensate the TPP0500B or TPP1000 probes .....	83
Compensate passive probes .....	85
Connect to a network (LAN) .....	86
Mount a network drive from a standard instrument .....	87
Mount a network drive from a Windows 10 instrument .....	88
Unmount a network drive from a standard instrument .....	89
Unmount a network drive from a Windows 10 instrument .....	89
Connect the oscilloscope to a PC using a USB cable .....	89
Remote access from a Web browser .....	90

Remote access to a Windows 10 instrument .....	92
Install TightVNC on the Windows 10 instrument .....	92
Install TightVNC on a PC .....	93
Deskew analog input channels - quick visual method .....	94
Deskew analog input channels - measurement method .....	95
Connect a keyboard or mouse .....	96
Connect an external monitor or projector .....	96
Prevent ESD Guidelines .....	97

## Analog channel operating basics

Acquiring a signal .....	99
Quickly display a waveform (Autoset) .....	100
Set Horizontal parameters .....	101
How to trigger on a signal .....	102
Set the acquisition mode .....	104
Start and stop an acquisition .....	105
Add a channel waveform to the display .....	105
Configure channel and waveform settings .....	106
Add a math, reference, or bus waveform .....	111
Add a measurement .....	112
Configure a measurement .....	115
Delete a Measurement or Search badge .....	116
Add a plot of a measurement .....	117
Display a Histogram plot .....	119
Display a Time Trend plot .....	119
Display a Spectrum plot .....	119
Display an XY or XYZ plot .....	120
Display an Eye Diagram plot .....	120
Add Mask Testing to an Eye Diagram plot .....	121
Display an FFT math waveform .....	122
Display a B-H curve plot for Magnetic Property measurement .....	123
Display an I vs (integral of) V plot for I vs V measurement .....	123
Display an Inductance Curve plot for Inductance measurement .....	123
Display a Safe Operating Area (SOA) measurement and plot .....	124
Display a power Switching Loss (SWL) Trajectory plot .....	124
Display instantaneous power and energy plots from the Power Quality measurement .....	125
Display a Harmonics Bar Graph plot for the Power Harmonics measurement .....	125
Display a Power Supply Rejection Ratio plot .....	126
Display a Control Loop Response (Bode) plot .....	126

Add a Search .....	127
Change waveform view settings .....	129
Display and configure cursors .....	130
Using Default Setup .....	132
Using Fast Acq .....	132
Add a note to a view .....	134
Delete a Note .....	135

## Acquiring digital signals

Connect and set up digital signals .....	137
Add a serial bus to the Waveform view .....	139
Add a parallel bus to the Waveform view .....	141

## Advanced triggering

Triggering concepts .....	145
Trigger on a pulse width event .....	146
Trigger on a timeout event .....	147
Trigger on a runt signal event .....	148
Trigger on a signal window event .....	148
Trigger on a logic event .....	149
Trigger on a setup-hold event .....	150
Trigger on a rise or fall time event .....	150
Set up trigger on a parallel bus .....	151
Set up trigger on a serial bus .....	151
Trigger on sequential events (A and B triggers) .....	152
Set Trigger Holdoff .....	153
Trigger on an external signal using the AUX input (MSO58LP, MSO64 only) .....	154
Create Visual Trigger areas .....	154
Edit visual trigger areas on the screen .....	155
Edit visual trigger areas using the Area menu .....	158

## Using Spectrum View

Spectrum View concepts .....	163
Display a spectrum trace .....	166
The Spectrum View user interface .....	167
Spectrum View and Spectrum Time .....	169
Spectrum trace handle and trace types .....	172
Spectrum peak markers .....	173

Spectrum View and front panel controls .....	174
Spectrum View and badges .....	175
Save and recall spectrum traces .....	177

## Setting waveform display parameters

Set waveform display mode (Stacked or Overlay) .....	179
Set the Waveform Interpolation mode .....	180
Set the waveform persistence, style, and intensity .....	180
Set the graticule style and intensity .....	181

## Zooming on waveforms

Turn on Zoom mode .....	183
Using Wave Inspector front-panel controls for zoom .....	185
Zoom mode and Searches .....	185

## Customizing measurements

Label a measurement .....	187
Set measurement reference levels .....	188
Set measurement gates .....	190
Set measurement filters .....	191
Set measurement limits .....	192

## Saving and recalling information

Save a screen image .....	193
Save a waveform or spectrum trace to a file .....	194
Save instrument settings to a file .....	195
Save reports .....	196
Save sessions .....	197
Recall a reference waveform or spectrum trace .....	198
Recall a Setup file .....	200
Recall a Session file .....	201
Open a mask test file for an Eye Diagram plot .....	202

## Menus and dialog boxes

The Acquisition configuration menu .....	203
Save On Trigger menu .....	205

FastFrame Panel .....	206
FastFrame badge .....	207
FastFrame badge configuration menu .....	209
Add Measurements configuration menu overview .....	210
The Standard measurements tab .....	211
Amplitude Measurements panel .....	212
Timing Measurements panel .....	214
Jitter Measurements panel .....	217
The Jitter tab (Advanced Jitter and Eye Analysis) (option 5-DJA, 6-DJA) .....	217
The Power tab (optional) .....	222
The DDR measurements tab (Requires option SUP6-DBDDR3) .....	226
Measurement configuration menu overview .....	228
Measurement Name panel (Measurement configuration menu) .....	230
Configure panel (Measurement configuration menu) .....	230
Reference Levels panel (Measurement configuration menu) .....	231
Clock Recovery panel (Measurement configuration menu) .....	234
Clock Recovery- Advanced Settings configuration menu .....	239
Gating panel (Measurement configuration menu) .....	240
Filter/Limit Results panel (Measurement Settings menu) .....	241
Power measurement configuration menu overview (optional) .....	242
Power Measurement Name panel (Measurement configuration menu) .....	243
Configure panel (Power measurement configuration menu) .....	244
Connection setup for Frequency Response Analysis (FRA) measurements .....	258
Configure Profile .....	260
Power Autoset .....	261
Power Preset .....	262
SOA Mask definition controls and fields .....	263
Save Mask menu (SOA power measurement) .....	264
Recall Mask menu (SOA power measurement) .....	265
Reference Levels panel (Power measurement configuration Menu) .....	268
Gating panel (Power measurement configuration menu) .....	268
DDR measurement configuration menu overview (Requires option SUP6-DBDDR3) .....	269
DDR measurement name panel (Measurement configuration menu) .....	270
Configure panel (DDR measurement configuration menu) .....	271
Reference Levels panel (DDR measurement configuration Menu) .....	271
Gating panel (DDR measurement configuration menu) .....	272
Filter/Limit results panel (DDR measurement configuration Menu) .....	272
Bus configuration menu .....	272
ARINC 429 serial bus menu .....	274



---

Audio serial bus configuration menu .....	275
CAN serial bus configuration menu .....	277
Ethernet serial bus menu .....	280
FlexRay serial bus configuration menu .....	282
I2C serial bus configuration menu .....	284
I3C serial bus configuration menu .....	286
LIN serial bus configuration menu .....	288
MIL-STD-1553 serial bus menu .....	290
Parallel Bus configuration menu .....	292
Parallel Bus - Define Inputs menu .....	294
RS232 serial bus menu .....	295
SENT serial bus configuration menu .....	297
SPI serial bus configuration menu .....	299
SPMI serial bus configuration menu .....	301
USB serial bus configuration menu .....	303
Add Plot configuration menu .....	306
Add Results Table configuration menu .....	307
Results Tables operations overview .....	308
Measurement Table configuration menu .....	308
Save As configuration menu (Measurement Results Table) .....	310
Search Results table menu .....	311
Bus Decode Results table configuration menu .....	313
Save As configuration menu (Bus Decode Results table) .....	314
Harmonics/Custom Results table configuration menu (option 5-PWR, 6-PWR only) .....	315
Save As configuration menu (Harmonics/Custom Results Table) .....	316
Annotation and navigation on waveform plots/data and results table (optional) .....	318
Navigation on Bar Graph and Harmonics Results Table (optional) .....	319
Search configuration menu overview .....	319
Bus Search configuration menus .....	320
ARINC 429 serial bus search configuration menu .....	321
Audio serial bus search configuration menu .....	323
CAN serial bus search configuration menu (when not using a .dbc symbol definition file) .....	324
CAN serial bus search configuration menu (when using a .dbc symbol definition file) .....	326
CAN DBC Symbolic Configuration menu .....	328
Ethernet serial bus search configuration menu .....	328
FlexRay serial bus search configuration menu .....	331
I2C serial bus search configuration menu .....	333
I3C serial bus Search configuration menu .....	334

LIN serial bus search configuration menu .....	344
MIL-STD-1553 serial bus search configuration menu .....	345
Parallel bus search configuration menu .....	347
RS-232 serial bus search configuration menu .....	347
SENT serial bus search configuration menu .....	348
SPI serial bus search configuration menu .....	351
SPMI serial bus search configuration menu .....	351
USB serial bus search configuration menu .....	353
Edge Search configuration menu .....	355
Logic search configuration menu .....	356
Logic Search - Define Inputs configuration menu .....	359
Pulse Width Search configuration menu .....	359
Rise/Fall Time Search configuration menu .....	361
Runt Search configuration menu .....	363
Setup and Hold Search configuration menu .....	365
Setup and Hold Search - Define Inputs configuration menu .....	367
Timeout Search configuration menu .....	368
Window Search configuration menu .....	369
DDR Read Search configuration menu (Requires option SUP6-DBDDR3) .....	372
DDR Write Search configuration menu (Requires option SUP6-DBDDR3) .....	374
DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3) .....	376
DQ/DQS Levels Configure menu .....	378
DDR Input Configure menu .....	379
Analog Channel configuration menu .....	379
Probe Setup panel (Channel configuration menu) .....	380
Unsupported Probe dialog .....	382
Probe Compensation configuration menu (analog channels Probe Setup panel) .....	382
Spectrum View panel (Channel configuration menu) .....	383
Other panel (Channel configuration menu) .....	385
Deskew configuration menu (Other panel, Channel configuration menu) .....	386
AFG configuration menu .....	388
Cursor configuration menu .....	390
Date and Time configuration menu .....	391
Digital channel configuration menu .....	392
Draw A Box Menu .....	393
DVM configuration menu .....	393
Horizontal configuration menu .....	394
Mask Badge configuration menu .....	397

---

Math configuration menu overview .....	397
Math configuration menu .....	397
Equation Editor (Math configuration menu) .....	401
Add Filter menu (math Equation Editor) .....	402
Add Variable menu (math Equation Editor) .....	403
Add Functions (math Equation Editor) .....	403
Menu bar overview .....	406
Recall configuration menu (File menu) .....	407
Save As configuration menu (File menu) .....	410
Browse Save As Location configuration menu .....	414
File Utilities configuration (File menu) .....	416
Mount Network Drive configuration menu .....	419
Undo, Redo (Edit menu) .....	420
Application (Menu bar) .....	420
User Preferences (Utility menu) .....	420
Define Custom Colors menu .....	424
I/O (Utility menu) .....	426
LAN Reset configuration menu (Utility > I/O menu) .....	429
Self Test configuration menu (Utility menu) .....	430
Calibration configuration menu (Utility menu) .....	431
Security configuration menu for standard instruments (Utility menu) .....	432
TekSecure Erase Memory .....	433
Security configuration menu (optional) .....	433
Enter Password configuration menu (optional) .....	435
Set Password configuration menu (optional) .....	436
Change Password configuration menu (optional) .....	436
Demo (Utility menu) .....	438
Help... (Help menu) .....	438
User Interface Tutorial (Help menu) .....	438
About (Help menu) .....	439
Location to Save Exit Key configuration menu .....	439
Browse License Files menu (Help > About) .....	441
Font Color menu (Text Settings configuration) .....	443
Text Settings configuration menu (Note and Waveform labels text) .....	443
Plot configuration menus .....	444
Eye Diagram plot configuration menu .....	444
Recall Mask File configuration menu .....	447
FastFrame Timestamp Trend plot configuration menu .....	449
Math FFT plot configuration menu (Math waveform) .....	450

Histogram plot configuration menu .....	454
Bathtub plot configuration menu .....	455
Spectrum plot configuration menu .....	457
Trend Plot configuration menu .....	459
Plot XY configuration menu .....	460
XYZ plot configuration menu .....	461
Harmonics Bar Graph plot configuration menu (optional) .....	462
Inductance Curve configuration menu (Magnetic Analysis power measurement) (optional) .....	463
B-H curve configuration menu (Magnetic Analysis power measurement) (optional) .....	464
I vs (integral of) V plot configuration menu (Magnetic Analysis power measurement) (optional) .....	465
Control Loop Response (Bode) plot configuration menu .....	466
Power Supply Rejection Ratio plot configuration menu .....	468
Power plots and cursors .....	469
SOA plot configuration menu (optional) .....	470
Trajectory plot configuration menu (Switching Loss power measurement) (optional) .....	471
Save As configuration menu (plot Save panel, Save Plot Image button) .....	472
Save As configuration menu (plot Save panel, Save Plot Data button) .....	473
Reference waveform configuration menu .....	474
Recall configuration menu (Ref waveform configuration menu) .....	475
Search configuration menu .....	477
Spectrum View menus .....	478
Spectrum View window configuration menu .....	478
Selected Trace configuration menu (Spectrum View) .....	480
Spectrum View Cursors menu .....	480
Spectrum badge menu .....	482
Trigger configuration menu overview .....	485
Edge Trigger configuration menu .....	485
Bus Trigger configuration .....	488
Logic Trigger configuration menu .....	527
Logic Trigger - Define Inputs configuration menu .....	529
Pulse Width Trigger configuration menu .....	530
Rise/Fall Time Trigger configuration menu .....	532
Runt Trigger configuration menu .....	535
Sequence Trigger configuration menu .....	538
A Trigger Event configuration menu .....	541
B Trigger Event configuration menu .....	541
Setup and Hold Trigger configuration menu .....	542

Setup and Hold Trigger - Define Inputs configuration menu .....	545
Timeout Trigger configuration menu .....	545
Logic Qualification - Define Inputs configuration menu .....	548
Window Trigger configuration menu .....	549
Visual Trigger panel .....	553
Virtual Keyboard .....	554
Bus trigger radix-specific virtual keypads .....	554
Virtual Keypad .....	555
Visual Trigger Area configuration menu .....	555
Visual Trigger Area Combinatorial Logic menu .....	558
Right click menu functions associated with visual trigger areas .....	559
Waveform View configuration menu .....	559

## Waveform acquisition concepts

Acquisition concepts .....	563
Acquisition hardware .....	563
Sampling process .....	563
Real-Time sampling .....	563
Interpolated Real-Time sampling .....	564
Waveform record .....	564
Interpolation .....	564
Acquisition modes .....	565
How the acquisition modes work .....	565
FastFrame concepts .....	568
Waveform sample interpolation .....	569
Coupling .....	569
Scaling and positioning .....	570
Vertical acquisition considerations .....	571
Horizontal acquisition considerations .....	572

## Trigger concepts

Triggering concepts .....	573
Trigger sources .....	574
Trigger types .....	575
Trigger modes .....	576
Trigger holdoff .....	576
Trigger coupling .....	577
Trigger slope and level .....	578

Trigger position in waveform record .....	578
Trigger delay .....	579
Advanced triggering .....	579
Bus triggering concepts .....	579
Pulse width trigger concepts .....	580
Timeout trigger .....	580
Runt trigger .....	580
Window trigger .....	581
Logic trigger concepts .....	581
Setup and Hold trigger concepts .....	581
Rise/Fall time trigger concepts .....	582
Sequential (A B) trigger concepts .....	582
Visual Trigger concepts .....	583

## Waveform display concepts

Waveform display overview .....	585
Waveform preview mode .....	586
Horizontal position and the horizontal reference point .....	586

## Measurement concepts

Measurement variables .....	587
Missing or out-of-range samples .....	590
Math waveforms .....	591
Math waveform elements .....	593
Math waveform sources .....	593
Guidelines for working with math waveforms .....	594
Math waveform editor syntax .....	595
Math waveform differentiation .....	596
Math waveform offset, position, and scale .....	597
Waveform integration .....	597
FFT process .....	598
FFT and aliasing .....	599
Blackman-Harris FFT window concepts .....	600
Flatop2 window .....	601
Gaussian window .....	602
Hanning FFT window .....	603
Hamming window .....	604
Kaiser-Bessel FFT window .....	605

Rectangular window .....	606
Tek-Exponential window .....	607

## Measurement algorithms

Amplitude measurement algorithms .....	609
AC RMS measurement algorithm .....	609
Area measurement algorithm .....	609
Amplitude measurement algorithm .....	609
Base measurement algorithm .....	609
Integration algorithm .....	610
Maximum measurement algorithm .....	611
Mean measurement algorithm .....	611
Minimum measurement algorithm .....	611
Negative Overshoot measurement algorithm .....	611
Positive Overshoot measurement algorithm .....	611
Peak-To-Peak measurement algorithm .....	612
RMS measurement algorithm .....	612
Top measurement algorithm .....	612
Timing measurement algorithms .....	612
Burst Width measurement algorithm .....	612
Data Rate measurement algorithm .....	612
Delay measurement algorithm .....	612
Falling slew rate .....	612
Fall Time measurement algorithm .....	613
Frequency measurement algorithm .....	614
High Time measurement algorithm .....	614
Hold Time measurement algorithm .....	614
Low Time measurement algorithm .....	615
N-Periods Duration measurement algorithm .....	615
Negative Duty Cycle measurement algorithm .....	616
Negative Pulse Width measurement algorithm .....	616
Period measurement algorithm .....	616
Phase measurement algorithm .....	616
Positive Duty Cycle measurement algorithm .....	617
Positive Pulse Width measurement algorithm .....	617
Rising Slew Rate measurement algorithm .....	617
Rise Time measurement algorithm .....	618
Setup .....	619
Skew .....	619

Time Outside Level measurement algorithm .....	620
Unit Interval measurement algorithm .....	620
Jitter measurement algorithms .....	621
AC Common Mode .....	621
Bit Amplitude measurement algorithm .....	621
Bit High measurement algorithm .....	621
Bit Low measurement algorithm .....	621
DC Common Mode measurement algorithm .....	621
Differential Crossover measurement algorithm .....	621
SSC Freq Dev measurement algorithm .....	622
SSC Modulation Rate measurement algorithm .....	622
TIE .....	622
T/nT Ratio measurement algorithm .....	623
DCD .....	623
DDJ .....	623
DJ .....	623
Dual Dirac deterministic jitter .....	623
F/2 measurement algorithm .....	623
F/4 measurement algorithm .....	624
F/8 measurement algorithm .....	624
J2 .....	624
J9 .....	624
Jitter Summary measurement .....	624
NPJ .....	624
Phase noise .....	624
PJ .....	625
RJ .....	625
RJ 66 measurement algorithm .....	625
SRJ .....	625
TJ@BER .....	625
Eye measurement algorithms .....	626
Eye Height measurement algorithm .....	626
Eye high .....	626
Eye low .....	627
Eye Width measurement algorithm .....	627
Height@BER .....	627
Q-factor .....	628
Width@BER .....	628



---

Power measurements: Input Analysis algorithms .....	628
Power Quality measurement algorithm .....	629
Harmonics algorithm .....	631
Input Capacitance algorithm .....	634
Inrush Current algorithm .....	634
Power measurements: Amplitude Analysis algorithms .....	635
Cycle Base measurement algorithm .....	635
Cycle Maximum measurement algorithm .....	635
Cycle Minimum measurement algorithm .....	635
Cycle Pea-to-Peak measurement algorithm .....	635
Cycle Top measurement algorithm .....	635
Cycle Amplitude measurement algorithm .....	635
Power measurements: Timing Analysis algorithms .....	636
Frequency measurement algorithm .....	636
Negative Duty Cycle measurement algorithm .....	636
Negative Pulse Width measurement algorithm .....	636
Period measurement algorithm .....	636
Positive Duty Cycle measurement algorithm .....	636
Positive Pulse Width measurement algorithm .....	637
Power measurements: Switching Analysis algorithms .....	637
Switching Loss algorithm .....	637
dv/dt algorithm .....	641
di/dt algorithm .....	641
SOA algorithm .....	642
RDS(on) algorithm .....	642
Power measurements: Magnetic Analysis algorithms .....	642
Inductance algorithm .....	642
I vs (integral of) V algorithm .....	643
Magnetic Loss algorithm .....	643
Magnetic Property algorithm .....	643
Power measurements: Output Analysis algorithms .....	645
Line Ripple algorithm .....	645
Switching Ripple algorithm .....	645
Efficiency algorithm .....	646
Turn On Time .....	647
Turn Off Time .....	647
Power measurements: Frequency Response Analysis algorithms .....	647
Control loop response measurement algorithm .....	647
Power supply rejection ratio measurement algorithm .....	648

DDR Amplitude Measurement algorithms .....	648
AOS (DDR) .....	648
AOS Per tCK (DDR) .....	648
AOS Per UI (DDR) .....	648
AUS (DDR) .....	648
AUS Per tCK (DDR) .....	649
AUS Per UI (DDR) .....	649
Vix(ac) (DDR) .....	649
DDR Time Measurement algorithms .....	649
tRPRE (DDR) .....	649
tWPRE (DDR) .....	649
tPST (DDR) .....	649
Hold Diff (DDR) .....	649
Setup Diff (DDR) .....	650
tCK(avg) (DDR) .....	650
tCH(avg) (DDR) .....	650
tCL(avg) (DDR) .....	651
tCH(abs) (DDR) .....	651
tCL(abs) (DDR) .....	651
tJIT(duty) (DDR) .....	652
tJIT(per) (DDR) .....	652
tJIT(cc) (DDR) .....	652
tERR(n) (DDR) .....	653
tERR(m-n) (DDR) .....	653
tDQSCK (DDR) .....	653
tCKSRE .....	654
tCKSRX .....	654
tCMD-CMD .....	654

## References

Power badge error and warning messages (optional) .....	655
---	-----

---

# TEKTRONIX SOFTWARE LICENSE AGREEMENT

THE PROGRAM, OR PROGRAMS, ENCODED OR INCORPORATED WITHIN EQUIPMENT OR ACCOMPANYING THIS AGREEMENT, IS FURNISHED SUBJECT TO THE TERMS AND CONDITIONS OF THIS AGREEMENT. RETENTION OF THE PROGRAM FOR MORE THAN THIRTY DAYS OR USE OF THE PROGRAM IN ANY MANNER WILL BE CONSIDERED ACCEPTANCE OF THE AGREEMENT TERMS. IF THESE TERMS ARE NOT ACCEPTABLE, THE UNUSED PROGRAM AND ANY ACCOMPANYING DOCUMENTATION SHOULD BE RETURNED PROMPTLY TO TEKTRONIX FOR A FULL REFUND OF THE LICENSE FEE PAID. (FOR INFORMATION REGARDING THE RETURN OF PROGRAMS ENCODED OR INCORPORATED WITHIN EQUIPMENT, CONTACT THE NEAREST TEKTRONIX SALES OFFICE.)

## DEFINITIONS.

"Tektronix" means Tektronix, Inc., an Oregon corporation, or local Tektronix legal entity that is supplying the equipment.

"Program" means the Tektronix software product (executable program and/or data) enclosed with this Agreement or included within the equipment with which this Agreement is packed.

"Customer" means the person or organization in whose name the Program was ordered.

## LICENSE.

Customer may:

1. Use the Program on a single machine at any one time; and
2. Copy the Program for archival or backup purposes, provided that no more than one (1) such copy is permitted to exist at any one time.

Each copy of the Program made by Customer must include a reproduction of any copyright notice or restrictive rights legend appearing in or on the copy of the Program as received from Tektronix.

Customer may not:

1. Use the Program on more than one machine at any one time;
2. Transfer the Program to any person or organization outside of Customer or the corporation of which Customer is a part without the prior written consent of Tektronix, except in connection with the transfer of the equipment within which the programs are encoded or incorporated;
3. Export or re-export, directly or indirectly, the program, any associated documentation, or the direct product thereof, to any country to which such export or re-export is restricted by law or regulation of the United States or any foreign government having jurisdiction without the prior authorization, if required, of the Office of Export Administration, Department of Commerce, Washington, D.C. and the corresponding agency of such foreign government;

4. Modify the Program, create derivative works, or merge the Program with another program;
5. Copy the documentation accompanying the Program; or
6. Decompile, decrypt, disassemble, or otherwise attempt to derive the source code, techniques, processes, algorithms, know-how, or other information (collectively "Reverse Engineer") from the Program or permit or induce any third party to do so. If however, any directly applicable law prohibits enforcement of the foregoing, Customer may engage in Reverse Engineering solely to obtain information necessary to achieve interoperability of independently created software with the Program, or as otherwise and to the limited extent permitted by directly applicable law, but only if Reverse Engineering is strictly necessary to obtain such information. If the Program contains any technology from a third-party that explicitly permits Reverse Engineering in its license, Customer may Reverse Engineer such third-party technology only to the limited extent permitted by that third-party license.

Title to the Program and all copies thereof, but not the media on which the Program or copies may reside, shall be and remain with Tektronix or others for whom Tektronix has obtained a respective licensing right.

Customer shall pay when due all property taxes that may now or hereafter be imposed, levied or assessed with respect to the possession or use of the Program or this license and shall file all reports required in connection with such taxes.

If the Program or any related documentation is acquired by or for an agency of the U.S. Government, the Program and documentation shall be considered "commercial computer software" or "commercial computer software documentation" respectively, as those terms are used in 48 CFR Sec. 12.212, 48 CFR Sec. 227.7202, or 48 CFR Sec. 252.227-7014, and are licensed with only those rights as are granted to all other licensees as set forth in this Agreement.

**THE PROGRAM MAY NOT BE USED, COPIED, MODIFIED, MERGED, OR TRANSFERRED TO ANOTHER EXCEPT AS EXPRESSLY PERMITTED BY THESE TERMS AND CONDITIONS.**

**UPON TRANSFER OF ANY COPY, MODIFICATION, OR MERGED PORTION OF THE PROGRAM, THE LICENSE GRANTED HEREIN IS AUTOMATICALLY TERMINATED WITH RESPECT TO THE PARTY THAT TRANSFERS THE PROGRAM.**

**TERM.**

The license granted herein is effective upon acceptance by Customer, and shall remain in effect until terminated as provided herein. The license may be terminated by Customer at any time upon written notice to Tektronix. The license may be terminated by Tektronix or any third party from whom Tektronix may have obtained a respective licensing right if Customer fails to comply with any term or condition and such failure is not remedied within thirty (30) days after notice hereof from Tektronix or such third party. Upon termination by either party, Customer shall return to Tektronix or destroy, the Program and all associated documentation, together with all copies in any form.

#### LIMITED WARRANTY.

The Program is provided "as is" without warranty of any kind, either express or implied. Tektronix does not warrant that the functions contained in the Program will meet Customer's requirements or that the operation of the Program will be uninterrupted or error-free.

TEKTRONIX AND ITS VENDORS DISCLAIM ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE.

LIMITATION OF LIABILITY, IN NO EVENT SHALL TEKTRONIX, ITS RESELLERS, OR OTHERS FROM WHOM TEKTRONIX HAS OBTAINED A LICENSING RIGHT BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES ARISING OUT OF OR CONNECTED WITH CUSTOMER'S POSSESSION OR USE OF THE PROGRAM, EVEN IF TEKTRONIX OR SUCH OTHERS HAS ADVANCE NOTICE OF THE POSSIBILITY OF SUCH DAMAGES.

#### THIRD-PARTY DISCLAIMER.

Except as expressly agreed otherwise, third parties from whom Tektronix may have obtained a licensing right do not warrant the program, do not assume any liability with respect to its use, and do not undertake to furnish any support or information relating thereto.

#### GENERAL.

This Agreement contains the entire agreement between the parties with respect to the use, reproduction, and transfer of the Program. Neither this Agreement nor the license granted herein is assignable or transferable by Customer without the prior written consent of Tektronix.

This Agreement and the license granted herein shall be governed by the laws of the state of Oregon.

All questions regarding this Agreement or the license granted herein should be directed to the nearest Tektronix Sales Office.



---

# Welcome to the 5/6 Series MSO instrument help

This help supports 5 Series MSO (MSO54, MSO56, MSO58, MSO58LP) and 6 Series MSO (MSO64) instruments. See the following lists for information on the key features of each instrument.

## MSO54, MSO56, MSO58 Key features and benefits

- Bandwidths from 350 MHz to 2 GHz
- 4-, 6-, and 8-channel models with FlexChannel<sup>®</sup> inputs
- Each FlexChannel input is dual-purpose, letting you connect either an analog probe (TekVPI<sup>®</sup> or BNC) or an eight-channel digital probe (the TLP058 FlexChannel Logic Probe)
- Each FlexChannel can display 8 digital channels (with TLP058), an analog waveform, a spectrum trace, or both an analog and spectral view of the same channel at the same time with independent controls for each view
- Large 15.6" HD (1920 x 1080 pixel) capacitive touch-screen display
- User interface optimized for 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
- Spectrum View enables simple, intuitive frequency domain analysis, independent of time domain controls, to show a spectral trace for each channel
- FastFrame<sup>™</sup> 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 (the number of waveforms depends on available system memory)
- Integrated options include a 50 MHz arbitrary/function generator (AFG), and a Digital Voltmeter (DVM) and trigger frequency counter
- Automated compliance testing solutions add compliance measurement features. See the *Compliance Testing options* Help topic
- Advanced serial bus triggering and analysis options let you decode and trigger on industry standard buses. See the *Serial bus and trigger options* Help topic
- Measurement and analysis options provide additional measurement and analysis functions. See the *Advanced Power Analysis* and *Advanced Jitter Analysis* option Help topics

---

Compliance testing options require the optional Microsoft Windows 10 operating system.

## MSO58LP Key features and benefits

- Low profile instrument with no display, comes ready to install in a 2U rack mount space
- 1 GHz bandwidth
- 8 channels with FlexChannel<sup>®</sup> inputs
- Each FlexChannel input is dual-purpose, letting you connect either an analog probe (TekVPI<sup>®</sup> or BNC) or an eight-channel digital probe (the TLP058 FlexChannel Logic Probe)
- Each FlexChannel can display 8 digital channels (with TLP058), an analog waveform, a spectrum trace, or both an analog and spectral view of the same channel at the same time with independent controls for each view
- Maximum 6.25 GS/s sample rate
- 125 M points record length on all channels
- 500,000 waveforms/second maximum waveform capture rate
- Spectrum View enables simple, intuitive frequency domain analysis, independent of time domain controls, to show a spectral trace for each channel
- FastFrame<sup>™</sup> 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 (the number of waveforms depends on available system memory)
- Integrated options include a 50 MHz arbitrary/function generator (AFG), and a Digital Voltmeter (DVM) and trigger frequency counter
- Advanced serial bus triggering and analysis options let you decode and trigger on industry standard buses. See the *Serial bus and trigger options* Help topic
- Power and Jitter options provide additional measurement and analysis functions. See the *Advanced Power Analysis* and *Advanced Jitter Analysis* option Help topics

## MSO64 Key features and benefits

- Bandwidths from 1 GHz to 8 GHz
- 4 channels with FlexChannel<sup>®</sup> inputs
- Each FlexChannel input is dual-purpose, letting you connect either an analog probe (TekVPI<sup>®</sup> or BNC) or an eight-channel digital probe (the TLP058 FlexChannel Logic Probe)
- Each FlexChannel can display 8 digital channels (with TLP058), an analog waveform, a spectrum trace, or both an analog and spectral view of the same channel at the same time with independent controls for each view
- FlexChannel inputs are compatible with TekVPI<sup>®</sup> probes
- Large 15.6" HD (1920 x 1080 pixel) capacitive touch-screen display



- User interface designed to optimize touch screen use and quickly access key settings
- Stacked mode places each channel or waveform in its own horizontal 'slice' on the screen, allowing for cleaner signal viewing and measuring
- Maximum 25 GS/s sample rate
- 62.5 M points record length on all channels (optional 125 M and 250 M record lengths available)
- 30,000 waveforms/second maximum waveform capture rate
- Spectrum View enables simple, intuitive frequency domain analysis, independent of time domain controls, to show a spectral trace for each channel
- No set limit on the number of math, reference, and bus waveforms you can display (the number of waveforms depends on available system memory)
- Integrated optional features include a 50 MHz arbitrary function generator (AFG), and a DVM and trigger frequency counter
- Optional serial triggering features enable you to isolate protocol-level events of interest in common aerospace, audio, automotive, computer, and embedded serial buses. See the *Serial bus and trigger options* topic in the instrument embedded Help, or the *6 Series MSO Serial Triggering and Analysis Applications Datasheet* (Tektronix part number 48W-61353-x) for more information

5/6 Series MSO Help, version 20190123-21:00 for Firmware v1.14.x.



# Product documents and support

## Related documents

Use the related documents for more information on instrument functions, 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	<i>5 Series MSO (MSO54, MSO56, MSO58) Installation and Safety Manual</i> (This document, Tektronix part number 071-3514-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. A Russian language version is available to download from the Tektronix web site (Tektronix part number 077-1361-xx)  <i>5 Series and 6 Series MSO Help</i> (Tektronix part number 077-1303-xx; Printable version of the instrument Help that contains context-sensitive descriptions of all instrument functions; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
How to remotely control the instrument	<i>5 Series and 6 Series MSO Programmer Manual</i> (Tektronix part number 077-1305-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Instrument specifications and procedures to verify the instrument meets specifications	<i>5 Series MSO MSO54, MSO56, MSO58, MSO58LP Specifications and Performance Verification Technical Reference</i> (Tektronix part number 077-1306-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Instrument theory of operation, troubleshooting, disassembly, and replaceable parts	<i>5 Series MSO MSO54, MSO56, MSO58 Service Manual</i> (Tektronix part number 077-1307-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Installing the instrument in a rack	<i>RM5 Rack Mount Kit Instructions</i> (Tektronix part number 071-3523-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Using the TLP058 Logic Probe	<i>TLP058 FlexChannel<sup>®</sup> Logic Probe Instructions</i> (Tektronix part number 071-3515-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )

## MSO58LP documents

To learn about	Use this document
How to use instrument functions	<p><i>5 Series MSO MSO58LP Installation and Safety Manual</i> (this document, Tektronix part number 071-3568-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. A Russian language version is available to download from the Tektronix web site (Tektronix part number 077-1404-xx)</p> <p><i>5 Series and 6 Series MSO Help</i> (Tektronix part number 077-1303-xx; Printable version of the instrument Help; available at <a href="http://www.tektronix.com/downloads">www.tektronix.com/downloads</a>)</p>
How to remotely control the instrument	<i>5 Series and 6 Series MSO Programmer Manual</i> (Tektronix part number 077-1305-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Instrument specifications and procedures to verify the instrument meets specifications	<i>5 Series MSO MSO54, MSO56, MSO58, MSO58LP Specifications and Performance Verification Technical Reference</i> (Tektronix part number 077-1306-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Using the TLP058 Logic Probe	<i>TLP058 FlexChannel<sup>®</sup> Logic Probe Instructions</i> (Tektronix part number 071-3515-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Converting the instrument for benchtop use	<i>MSO58LP Bench Conversion Kit Instructions</i> (Tektronix part number 075-1102-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )

## 6 Series MSO documents

To learn about	Use this document
How to use instrument functions	<p><i>5 Series and 6 Series MSO Help</i> (Tektronix part number 077-1303-xx; Printable version of the instrument Help; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a>)</p> <p><i>6 Series MSO Installation and Safety Manual</i> (this document, Tektronix part number 071-3579-xx); standard accessory with the instrument. Single document with English, Japanese, and Simplified Chinese languages. A Russian language version is available to download from the Tektronix web site (Tektronix part number 077-1432-xx)</p>
How to remotely control the instrument	<i>5 Series and 6 Series MSO Programmer Manual</i> (Tektronix part number 077-1305-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Instrument specifications and procedures to verify the instrument meets specifications	<i>6 Series MSO Specifications and Performance Verification Technical Reference</i> (Tektronix part number 077-1461-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Instrument theory of operation, troubleshooting, disassembly, and replaceable parts	<i>6 Series MSO Service Manual</i> (Tektronix part number 077-1462-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Installing the instrument in a rack	<i>RM5 Rack Mount Kit Instructions</i> (Tektronix part number 071-3523-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )
Using the TLP058 Logic Probe	<i>TLP058 FlexChannel<sup>®</sup> Logic Probe Instructions</i> (Tektronix part number 071-3515-xx; available at <a href="http://www.tek.com/downloads">www.tek.com/downloads</a> )

## 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
<i>5 Series MSO (MSO54, MSO56, MSO58) Installation and Safety Manual</i>	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
<i>5 Series MSO MSO58LP Installation and Safety Manual</i>	1	071-3568-xx
Power cord	1	Depends on region
Calibration certificate	1	N/A
Report of factory installed licenses	1	N/A

**MSO64 standard accessories**

Item	Quantity	Tektronix part number
Installation and Safety Manual	1	071-3579-xx
TPP1000 Passive Voltage Probe (1 GHz bandwidth).	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

**Recommended accessories**

See the Tektronix Web site ([www.tek.com](http://www.tek.com)) for the latest information on recommended accessories for this product.

**MSO54, MSO56, MSO58 recommended accessories**

Accessory	Tektronix part number
8 channel general purpose logic probe. Includes accessory kit.	TLP058
Rackmount kit	RM5
Mini keyboard	119-7275-xx
TekVPI-to-TekProbe BNC adapter	TPA-BNC
Deskew pulse generator	TEK-DPG
Hard transit case	HC5
<i>5 Series MSO (MSO54, MSO56, MSO58) Service Manual as a PDF file, download from the Tektronix Web site</i>	077-1307-xx
<i>5 Series and 6 Series MSO Programmer Interface Manual as a PDF file, download from the Tektronix Web site</i>	077-1305-xx
<i>5 Series MSO (MSO54, MSO56, MSO58, MSO58LP) Specifications and Performance Verification Technical Reference as a PDF file, download from the Tektronix Web site</i>	077-1306-xx



**MSO58LP recommended accessories**

Accessory	Tektronix part number
8 channel general purpose logic probe with accessory kit	TLP058
Mini keyboard	119-7275-xx
TekVPI-to-TekProbe BNC adapter	TPA-BNC
Deskew pulse generator	TEK-DPG
Hard transit case	016-2139-xx
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
<i>5 Series MSO MSO58LP Service Manual</i> as a PDF file, download from the Tektronix Web site	077-1405-xx
<i>5 Series and 6 Series MSO Programmer Interface Manual</i> as a PDF file, download from the Tektronix Web site	077-1305-xx
<i>5 Series MSO (MSO54, MSO56, MSO58, MSO58LP) Specifications and Performance Verification Technical Reference</i> as a PDF file, download from the Tektronix Web site	077-1306-xx

**MSO64 recommended accessories**

Accessory	Tektronix part number
8 channel general purpose logic probe with accessory kit	TLP058
Rackmount kit	RM5
Mini keyboard	119-7275-xx
TekVPI-to-TekProbe BNC adapter	TPA-BNC
Hard transit case	HC5
<i>6 Series MSO (MSO64) Service Manual</i> as a PDF file, download from the Tektronix Web site	077-1462-xx
<i>5 Series and 6 Series MSO Programmer Interface Manual</i> as a PDF file, download from the Tektronix Web site	077-1305-xx
<i>6 Series MSO (MSO64) Specifications and Performance Verification Technical Reference</i> as a PDF file, download from the Tektronix Web site	077-1461-xx

## Recommended probes

See the Tektronix Web site ([www.tek.com](http://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
TAP4000	4 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
TCP0150	DC to 20 MHz, TekVPI 150 A AC/DC current probe
TDP0500	500 MHz TekVPI differential voltage probe with $\pm 42$ V differential input voltage
TDP1000	1 GHz TekVPI differential voltage probe with $\pm 42$ V differential input voltage
TDP1500	1.5 GHz TekVPI differential voltage probe with $\pm 8.5$ V differential input voltage
TDP3500	3.5 GHz TekVPI differential voltage probe with $\pm 2$ V differential input voltage
TDP4000	4 GHz TekVPI differential voltage probe with $\pm 2$ V differential input voltage
TDP7704	4 GHz TekVPI TriMode voltage probe with TekFlex connector technology
TDP7706	6 GHz TekVPI TriMode voltage probe with TekFlex connector technology
TDP7708	8 GHz TekVPI TriMode voltage probe with TekFlex connector technology
THDP0100	$\pm 6$ kV, 100 MHz TekVPI high-voltage differential probe
THDP0200	$\pm 1.5$ kV, 200 MHz TekVPI high-voltage differential probe
TMDP0200	$\pm 750$ 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	$\pm 50$ V, 1GHZ IsoVu probe 3 m
TIVM1L	$\pm 50$ V, 1GHZ IsoVu probe 10 m
TPP0500B	300 V, 500 MHz 10X passive probe
TPP1000	300 V, 1 GHz 10X passive probe
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

---

**NOTE.**

*The TRCP0300, TRCP0600, and TRCP3000 current probes are non TekVPI probes. These probes do not automatically set their probe attenuation factor when connected to the oscilloscope. Double-tap the Channel badge and use the configuration menu to set the probe attenuation and other parameters as needed. The Attenuation factor for TRCP0300, TRCP0600, and TRCP3000 current probes are 50X, 100X, and 500X respectively.*

---



# Options

## Bandwidth options

These options let you upgrade a purchased oscilloscope to a higher bandwidth.

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

### 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 1: 5 Series bandwidth upgrades**

Option name	Description	Notes
<b>SUP5-BW3T54</b>	Bandwidth upgrade from 350 MHz to 500 MHz on 4-channel models	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.
<b>SUP5-BW3T56</b>	Bandwidth upgrade from 350 MHz to 500 MHz on 6-channel models	
<b>SUP5-BW3T58</b>	Bandwidth upgrade from 350 MHz to 500 MHz on 8-channel models	
<b>SUP5-BW3T104</b>	Bandwidth upgrade from 350 MHz to 1 GHz on 4-channel models, plus 4 TPP1000 probes.	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.
<b>SUP5-BW3T106</b>	Bandwidth upgrade from 350 MHz to 1 GHz on 6-channel models, plus 6 TPP1000 probes.	
<b>SUP5-BW3T108</b>	Bandwidth upgrade from 350 MHz to 1 GHz on 8-channel models, plus 8 TPP1000 probes.	

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

**Table 2: 6 Series bandwidth upgrades**

Option name	Description	Notes
<b>SUP6-BW10T254</b>	Bandwidth upgrade from 1 GHz to 2.5GHz on 4-channel models.	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.
<b>SUP6-BW10T404</b>	Bandwidth upgrade from 1 GHz to 4 GHz on 4-channel models.	
<b>SUP6-BW10T604</b>	Bandwidth upgrade from 1 GHz to 6 GHz on 4-channel models.	
<b>SUP6-BW10T804</b>	Bandwidth upgrade from 1 GHz to 8 GHz on 4-channel models.	
<b>SUP6-BW25T404</b>	Bandwidth upgrade from 2.5 GHz to 4 GHz on 4-channel models.	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.
<b>SUP6-BW25T604</b>	Bandwidth upgrade from 2.5 GHz to 6 GHz on 4-channel models.	
<b>SUP6-BW25T804</b>	Bandwidth upgrade from 2.5 GHz to 8 GHz on 4-channel models.	
<b>SUP6-BW40T604</b>	Bandwidth upgrade from 4 GHz to 6 GHz on 4-channel models.	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.
<b>SUP6-BW40T804</b>	Bandwidth upgrade from 4 GHz to 8 GHz on 4-channel models.	
<b>SUP6-BW60T804</b>	Bandwidth upgrade from 6 GHz to 8 GHz on 4-channel models.	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.

## Record length options

These options let you upgrade a purchased oscilloscope to increase the record length memory. Increased record length memory lets you capture and analyze more waveform data points.

---

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

---

**Table 3: 5 Series options**

Option name	Description
<b>SUP5-RL-125M</b>	<p>Upgrade from 62.5 million record points per channel to 125 million record points per channel maximum.</p> <p>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.</p> <p>The option license can only be installed on the oscilloscope for which it was purchased.</p>



**Table 4: 6 Series options**

Option name	Description
<b>SUP6-RL-125M</b>	<p>Upgrade from 62.5 million record points per channel to 125 million record points per channel maximum.</p> <p>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.</p> <p>The option license can only be installed on the oscilloscope for which it was purchased.</p>
<b>SUP6-RL-250M</b>	<p>Upgrade from 62.5 million record points per channel to 250 million record points per channel maximum.</p> <p>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.</p> <p>The option license can only be installed on the oscilloscope for which it was purchased.</p>
<b>SUP6-RL-1T2</b>	<p>Upgrade from 125 million record points per channel to 250 million record points per channel maximum.</p> <p>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.</p> <p>The option license can only be installed on the oscilloscope for which it was purchased.</p>

## Microsoft Windows 10 operating system option

These options add the Microsoft Windows 10 operating system to your oscilloscope. 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 oscilloscope application.
- The 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 upgrade option

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

Upgrade option name	Description
<b>SUP5-WIN or SUP6-WIN</b>	<p>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 oscilloscope application.</p> <p>The option can be installed on any instrument; it is not tied to a specific instrument.</p>

## Arbitrary Function Generator (AFG) (optional)

These options add a 50 MHz AFG function to your oscilloscope.

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

Upgrade option name	Description
<b>SUP5-AFG or SUP6-AFG</b>	<p>Adds arbitrary and function generation capability to your instrument.</p> <p>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.</p> <p>The option license can only be installed on the oscilloscope for which it was purchased.</p>

## Enhanced instrument security (factory option)

Advanced instrument security option provides the highest level of instrument security. This option configures the oscilloscope hardware to easily declassify the oscilloscope. This option 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. This option must be ordered at the same time you order an instrument.

Install option name	Description
<b>5-SEC or 6-SEC</b>	<p>No user access to the internal m.2 storage to store or save any user data to the oscilloscope memory.</p> <p>Data can only be saved to or read from a USB storage device connected to the instrument, or through the programmable interface.</p> <p>No hard drive connection bracket on the bottom of the instrument prevents installing the optional Windows OS solid state drive.</p> <p>Password protection to enable/disable external USB Host, USB Device and Ethernet communication ports.</p> <p>Password protection to enable/disable firmware upgrades or downgrades.</p>

### 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 upgrade a purchased oscilloscope to add these options.

### Serial trigger upgrade options

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

Option name	Description	Notes
<b>SUP5-SRAERO</b> or <b>SUP6-SRAERO</b>	Aerospace serial triggering and analysis (ARINC 429, MIL-STD-1553)	<p>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.</p> <p>You can order a floating license that lets you use this option on any instrument (checked out to one instrument at a time), or an instrument-specific license option.</p> <p>The instrument-specific option license can only be installed on the oscilloscope for which it was purchased.</p>
<b>SUP5-SRAUDIO</b> or <b>SUP6-SRAUDIO</b>	Audio serial triggering and analysis (I <sup>2</sup> S, LJ, RJ, TDM)	
<b>SUP5-SRAUTO</b> or <b>SUP6-SRAUTO</b>	Automotive serial triggering and analysis (CAN, CAN FD, LIN, FlexRay)	
<b>SUP5-SRAUTOSEN</b> or <b>SUP6-SRAUTOSEN</b>	Automotive SENT (Single Edge Nibble Transmission) serial triggering and analysis	
<b>SUP5-SRCOMP</b> or <b>SUP6-SRCOMP</b>	Computer serial triggering and analysis (RS-232/422/485/UART)	
<b>SUP5-SRENET</b> or <b>SUP6-SRENET</b>	Ethernet serial triggering and analysis (10BASE-T, 100BASE-T)	
<b>SUP5-SREMBD</b> or <b>SUP6-SREMBD</b>	Embedded serial triggering and analysis (I <sup>2</sup> C, SPI)	
<b>SUP5-SRPM</b> or <b>SUP6-SRPM</b>	Mobile device SPMI (System Power Management Interface) serial triggering and analysis	
<b>SUP5-SRUSB2</b> or <b>SUP6-SRUSB2</b>	USB serial triggering and analysis (USB 2.0 LS, FS, HS)	

## Compliance testing options

Compliance testing options provide compliance testing and analysis of industry standard buses. You can upgrade a purchased oscilloscope to add these options.

### Compliance testing upgrade options

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

Option name	Description	Notes
<b>SUP5-CMAUTOEN</b> or <b>SUP6-CMAUTOEN</b>	Automotive Ethernet (100BASE-T1) compliance testing. Requires optional 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.  A single-instrument option license can only be installed on the oscilloscope for which it was purchased.
<b>SUP5-CMUSB2</b> or <b>SUP6-CMUSB2</b>	USB2.0 automated compliance test solution. Requires optional Microsoft Windows 10 operating system	

## Advanced Jitter Analysis option

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 upgrade an oscilloscope to add these options.

### Advanced Jitter Analysis upgrade option

This option can be ordered after purchasing the oscilloscope.

Upgrade option name	Description
<b>SUP5-DJA</b> or <b>SUP6-DJA</b>	<p>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.</p> <p>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.</p> <p>You can order a floating license that lets you use this option on any instrument (checked out to one instrument at a time), or an instrument-specific license option.</p> <p>The instrument-specific option license can only be installed on the oscilloscope for which it was purchased.</p>

## Advanced Power Analysis option

This option adds industry standard power measurements, as well as a summary table that displays the most common measurements. You can upgrade an oscilloscope to add this option.

**Power upgrade option** Order this option to upgrade your oscilloscope with power measurements.

Upgrade option name	Description
<b>SUP5-PWR or SUP6-PWR</b>	<p>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.</p> <p>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.</p> <p>You can order a floating license that lets you use this option on any instrument (checked out to one instrument at a time), or an instrument-specific license option.</p> <p>The instrument-specific option license can only be installed on the oscilloscope for which it was purchased.</p>

## DDR option

This option adds industry standard DDR measurements, as well as a summary table that displays the most common measurements. You can upgrade an oscilloscope to add this option.

Order this option to upgrade your oscilloscope with DDR measurements.

### Prerequisites

Requires Advanced Jitter Analysis option (SUP6-DJA)

Upgrade option name	Description
<b>SUP6-DBDDR3</b>	<p>Adds the DDR3 measurements, including Overshoot, Undershoot, Voltage crossover, Preamble timing, Postamble timing, Clock jitter timing, Slew rate, Hold time, Setup time, and more.</p> <p>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.</p> <p>You can order a floating license that lets you use this option on any instrument (checked out to one instrument at a time), or an instrument-specific license option.</p> <p>The instrument-specific option license can only be installed on the oscilloscope for which it was purchased.</p>

---

**NOTE.**

---

## 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
<b>A0</b>	North America Power Cord
<b>A1</b>	Universal EURO Power Cord
<b>A2</b>	United Kingdom Power Cord
<b>A3</b>	Australia Power Cord
<b>A5</b>	Switzerland Power Cord
<b>A6</b>	Japan Power Cord
<b>A10</b>	China Power Cord
<b>A11</b>	India Power Cord
<b>A12</b>	Brazil Power Cord
<b>A99</b>	No Power Cord or AC Adapter
<b>E1</b>	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
<b>T3</b>	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.
<b>T5</b>	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.
<b>R5</b>	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.
<b>C3</b>	Three-year calibration service. Includes traceable or functional verification where applicable, for recommended calibrations. Coverage includes the initial calibration plus two years calibration coverage.
<b>C5</b>	Five-year calibration service. Includes traceable or functional verification where applicable, for recommended calibrations. Coverage includes the initial calibration plus four years calibration coverage.
<b>D1</b>	Factory calibration data report for the instrument.
<b>D3</b>	Three years of calibration data reports (with Option C3)
<b>D5</b>	Five years of calibration data reports (with Option C5)
<b>R5DW</b>	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 (Node (instrument) locked or Floating) to enable specific instrument features. Option licenses provide advanced functions for specific standards or measurement requirements.

Prerequisites:

- A license file for each option. Contact Tektronix Customer Service to purchase and obtain option license file(s). Node Locked option licenses are specific to the oscilloscope serial number, and cannot be used on other oscilloscopes. Floating licenses can be used on any instrument, but can only be checked out to one instrument at a time.

License (.lic) files are downloaded from the Tektronix AMS tool at [www.tek.com/products/product-license](http://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 a Node Locked option license one time. If you need to reinstall an uninstalled Node Locked 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 26.

**Drive names for USB memory devices.**

Use the following 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
<b>MSO54, MSO56, MSO58, MSO64, without Windows OS</b>		
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)
<b>MSO58LP, without Windows OS</b>		
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)
<b>Instruments with Windows OS and USB port labels</b>		
Root drive	C	User-accessible memory on the oscilloscope.
USB ports	<b>Dynamic port letter assignment</b>	If Windows operating system is installed, 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.

## How to uninstall (return) an option license

Use this process to uninstall (return) an option license (Node Locked or Floating).

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.

Floating licenses can be used on any compatible instrument, but can only be checked out to use on one instrument at a time. This process removes the option license from the instrument so that you can check (return) it back into the Tektronix TekAMS system.

When you check out an option Floating license from your TekAMS account, you stipulate a time period for which the option license can be used. When the license expiration time is close, the instrument shows warning messages at 72 hours, 24 hours, and 1 hour prior to the expiration date.

At the expiration time, the TekAMS account automatically sets the license as available. The instrument gives you a four-hour grace period.

If you do nothing, the instrument automatically deactivates the license after the grace period expires and restarts the TekScope application (on instruments with Windows OS installed) or performs a warm reboot (base instrument). You should uninstall (return) the license key file before the expiration date to prevent a power cycle and possible disruption of measurements.

---

**NOTE.** *If you uninstall a Node Locked option license, you cannot reinstall the license using the same file used to install it. If you need to reinstall an uninstalled Node Locked option, contact Tektronix Customer Support to obtain a new option license file.*

---

To uninstall a license:

1. Insert a USB memory stick or device in an available USB port.
2. Select **Help > About**.
3. Tap the option license in the list that you want to uninstall (return to your Tektronix AMS account).
4. Tap the **Return License** button.

The oscilloscope opens the **Location to Save the Exit Key** menu.

5. Navigate to and select the location at which to save the license key uninstall file:
  - a. For an instrument not connected to the internet, or installed in a secure environment, save the license file to a USB memory device connected to the instrument. See the table at the end of this procedure for USB port names.
  - b. For an instrument with the Microsoft Windows 10 OS option that is connected to the internet, you can save the uninstall key file to a location on the instrument SSD drive or to a network drive.

6. Tap **Create**. The oscilloscope saves the license key file to the specified location and removes the license from the Installed Options list.
7. Power cycle the oscilloscope before taking any measurements.
8. From a PC with the license file USB memory device attached, or from a browser on an instrument connected to the internet, access the Tektronix AMS Web site. The site path was provided when you purchased the license and created a Tektronix AMS account.
9. Use the Tektronix AMS site to select and return the license key file to your Tektronix AMS account.

To install an option license, see [How to install an option license](#) on page 24.

**Drive names for system and USB memory devices.**

Use the following 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
<b>MSO54, MSO56, MSO58, MSO64, without Windows OS</b>		
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)
<b>MSO58LP, without Windows OS</b>		
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)
<b>Instruments with Windows OS and USB port labels</b>		
Root drive	C	User-accessible memory on the oscilloscope.
USB ports	<b>Dynamic port letter assignment</b>	If Windows operating system is installed, 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](http://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 standard accessories and ordered items. If you purchased factory installed options such as a Serial Bus and Triggering option, or the Power measurements option, tap **Help** > **About** to confirm that the option(s) are listed in the **Installed Options** table.

### MSO54, MSO56, MSO58 standard accessories

Item	Quantity	Tektronix part number
<i>5 Series MSO (MSO54, MSO56, MSO58) Installation and Safety Manual</i>	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

**MSO64 standard accessories**

Item	Quantity	Tektronix part number
Installation and Safety Manual	1	071-3579-xx
TPP1000 Passive Voltage Probe (1 GHz bandwidth).	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

## 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.

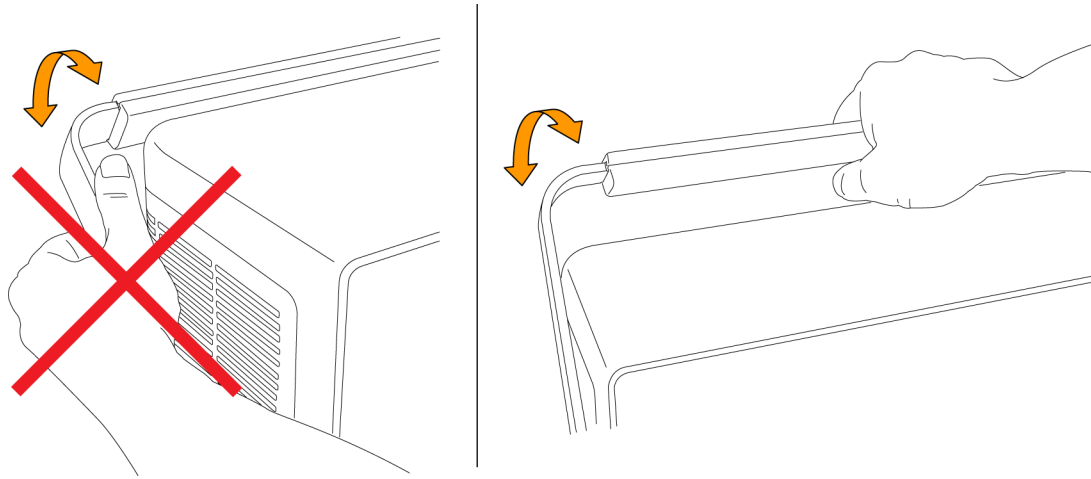


**CAUTION.** *Hold the top of the handle to rotate the handle on the instrument. Do not hold the handle from the sides and rotate, as this can pinch the base of your thumb between the handle and the case.*

*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), Noncondensing.  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 <sub>AC RMS</sub> , ±10%, single phase
Power source frequency	50/60 Hz, 100-240 V 400 Hz, 115 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 $\Omega$ setting, maximum input voltage at BNC	300 V <sub>RMS</sub> Transient Overvoltage is 0V. The measuring terminals on this product are not rated for connection to mains or Category II, III, or IV circuits. Measurement Category II
Analog input channels, 50 $\Omega$ setting, maximum input voltage at BNC	5 V <sub>RMS</sub> Transient Overvoltage is 0V. The measuring terminals on this product are not rated for connection to mains or Category II, III, or IV circuits.
Digital input channels, maximum input voltage range at digital inputs	Observe probe ratings TLP058; ±42 V <sub>P</sub>
<b>Ref In</b> maximum input voltage at BNC (rear panel)	7 V <sub>PP</sub>
AUX trigger input	±5 V <sub>RMS</sub>

## 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.

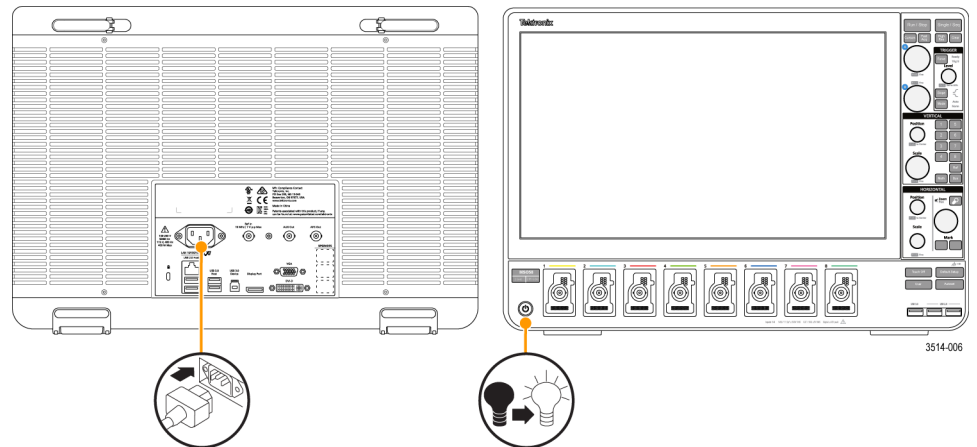


Figure 1: MSO54, MSO56, MSO58 power cord connector and power standby switch

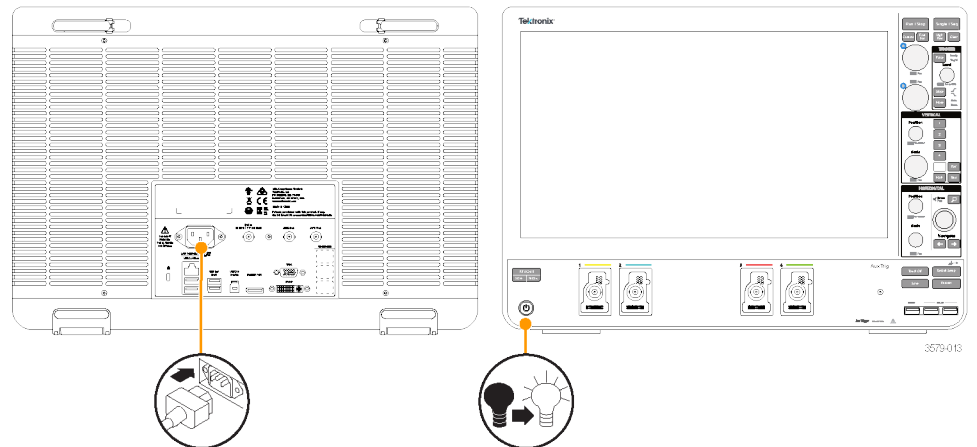


Figure 2: MSO6 Series power cord connector and power standby switch

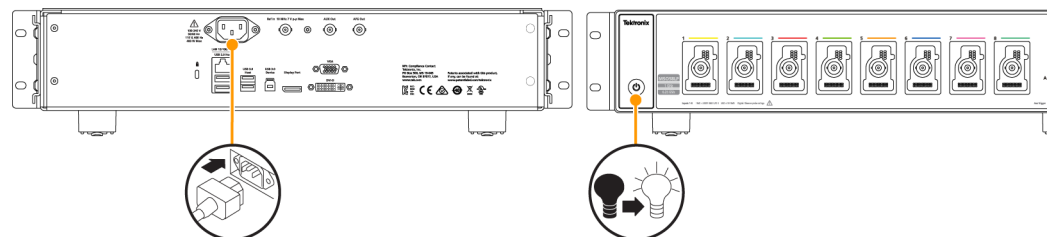


Figure 3: MSO58LP power cord connector and power standby switch

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:  
Unlit – 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 the 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 status of all power-on self tests are **Passed**.

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

- a. Power cycle the oscilloscope.
- b. Tap **Utility > Self Test**. If one or more power-on self tests still shows **Failed**, contact Tektronix Customer Support.

## Installing and activating Windows 10 option

Use the following instructions to install the Microsoft Windows 10 solid state drive (upgrade installed after purchasing the instrument) and activate Windows 10 on the instrument.

---

**NOTE.** *The Windows option cannot be installed in a Low Profile instrument.*

---

### Install the Windows SSD drive

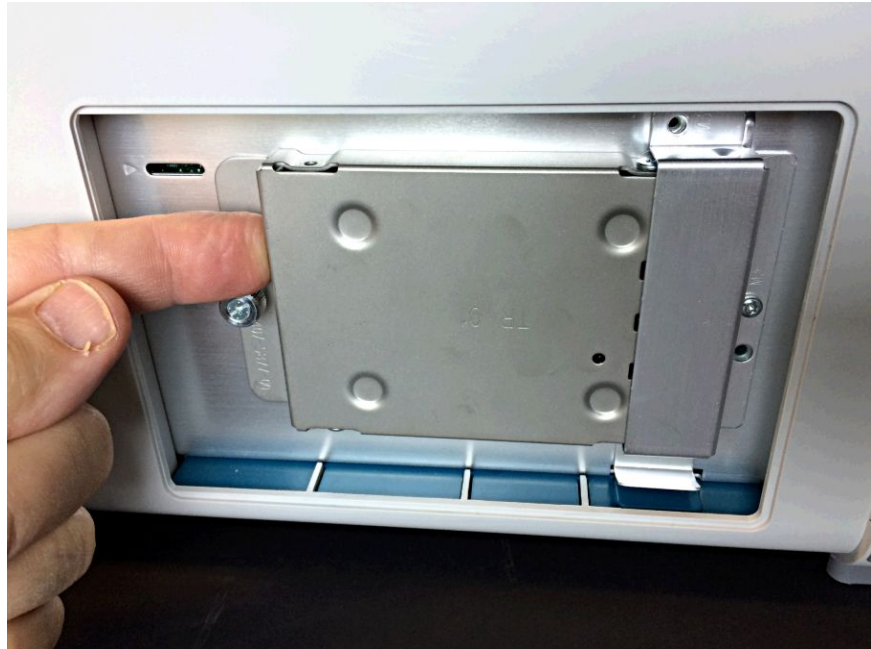
The optional SSD assembly installs in the bottom of the instrument.

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 [Powering on Windows for the first time](#) on page 36 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) Windows drive in an instrument. Once the 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 37).

## 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 Windows 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 38 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 38 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 38 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 Windows SSD from one instrument to another:
  - Windows should remain activated. To verify activation, see *Verifying Windows activation* on page 38. 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 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.
- You can use the Application menu to access installed applications.

### Updating the Windows TekScope application software

The Windows TekScope application does not automatically update when newer versions are released. You will need to manually download and install the newer software. The oscilloscope application cannot be running when installing the software.

To update the Windows version of the 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:**



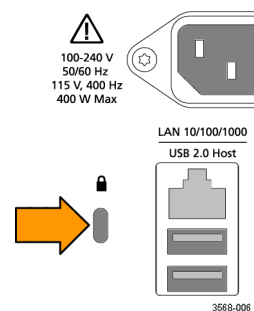
Follow the above instructions on a network-connected PC,

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.
5. Double-tap on the installation file to update the application software; follow any on-screen instructions.

## Secure (lock) the oscilloscope

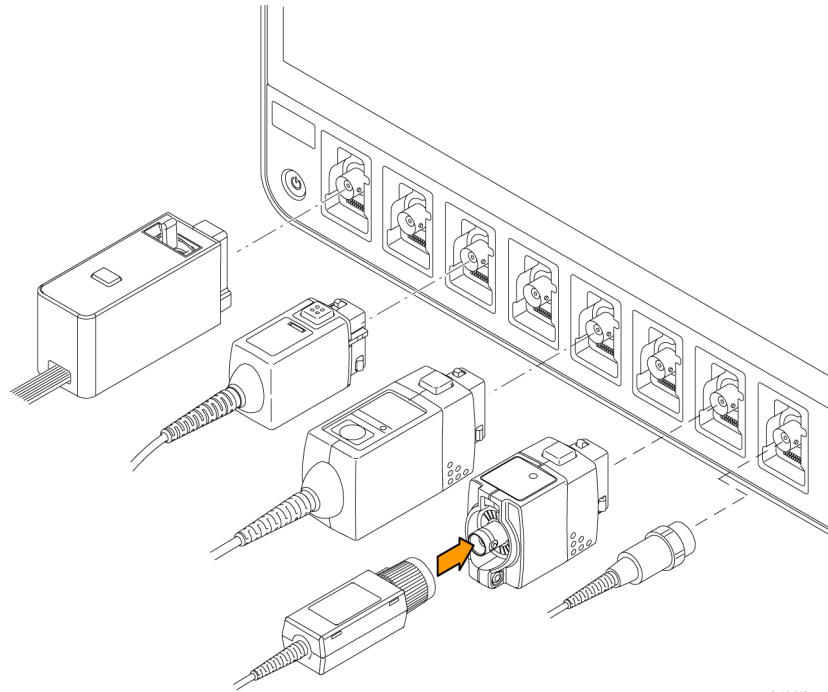
Lock an oscilloscope to a test bench or equipment rack to prevent property loss.

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



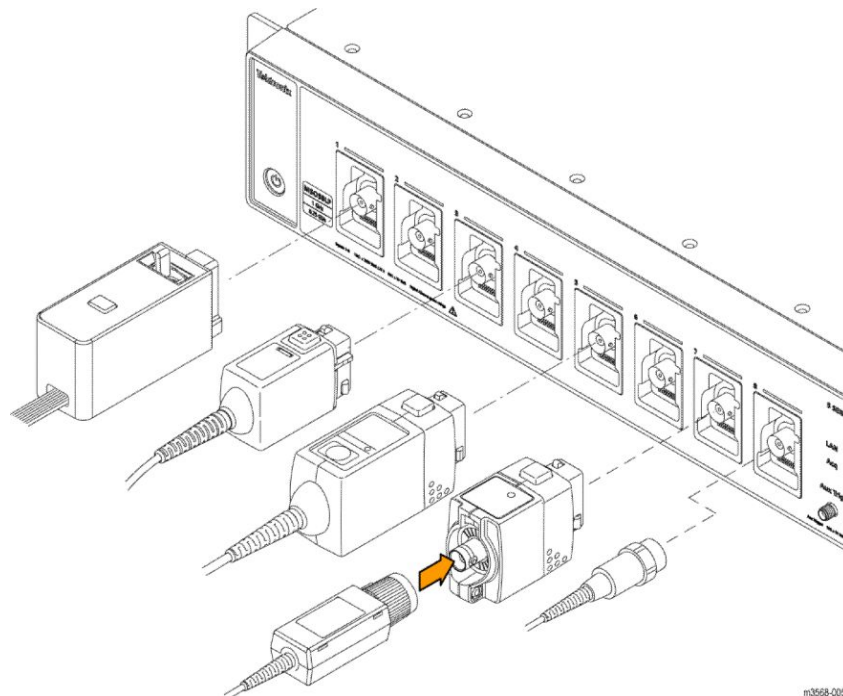
## Connecting Probes

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



3514-019

**Figure 4: Connecting probes to the MSO54, MSO56, or MSO58**



m3658-005

Figure 5: Connecting probes to the MSO58LP

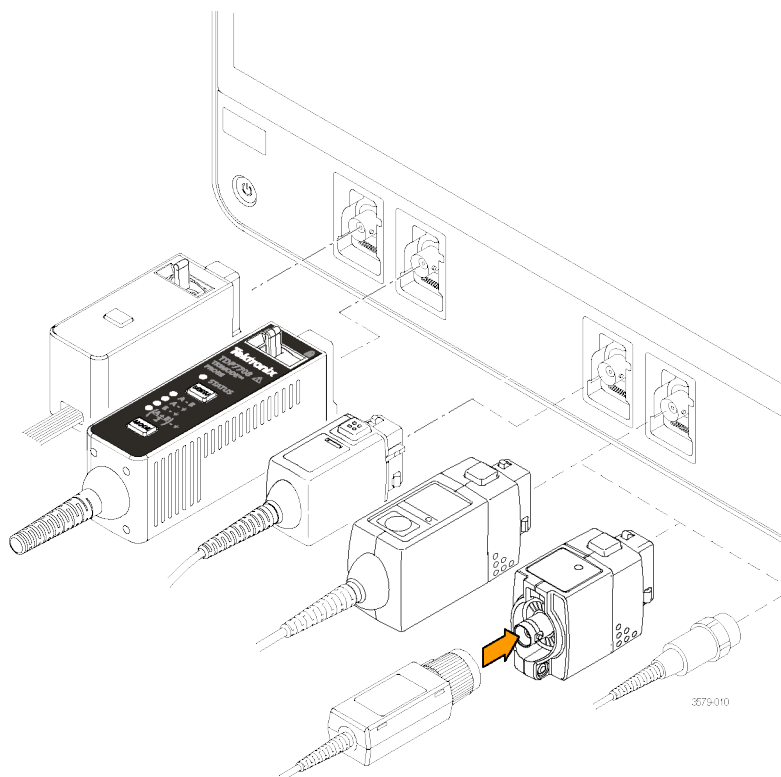


Figure 6: Connecting probes to the MSO6 Series

Connect TPP0500B, 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 or a TDP7700 Series TriMode™ 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 connector 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.

---

**NOTE.** *Connecting a probe does not automatically enable that channel (make it active). Use the instrument controls or programmatic interface to turn on a channel and 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 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.

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

Make sure to allow at least two inches of clearance on the sides for air ventilation, and on the back for any cables you attach to the rear panel.

The MSO58LP comes equipped with rack mount hardware installed so that you can immediately 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.

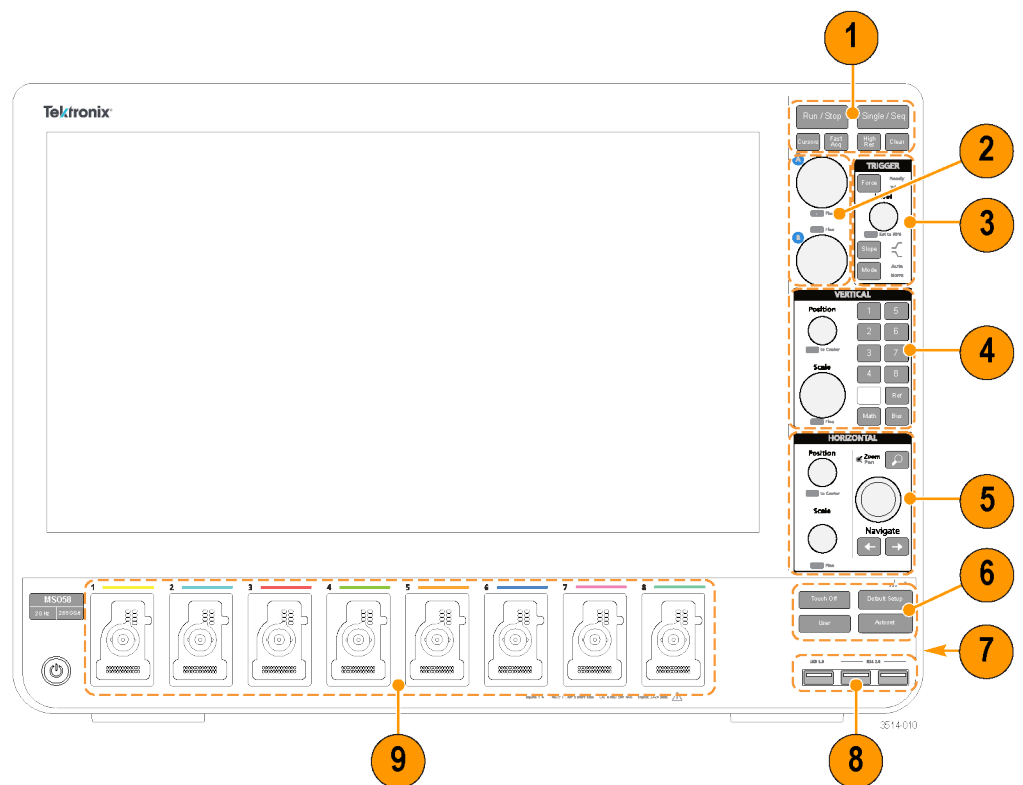


Figure 7: 5 Series MSO controls

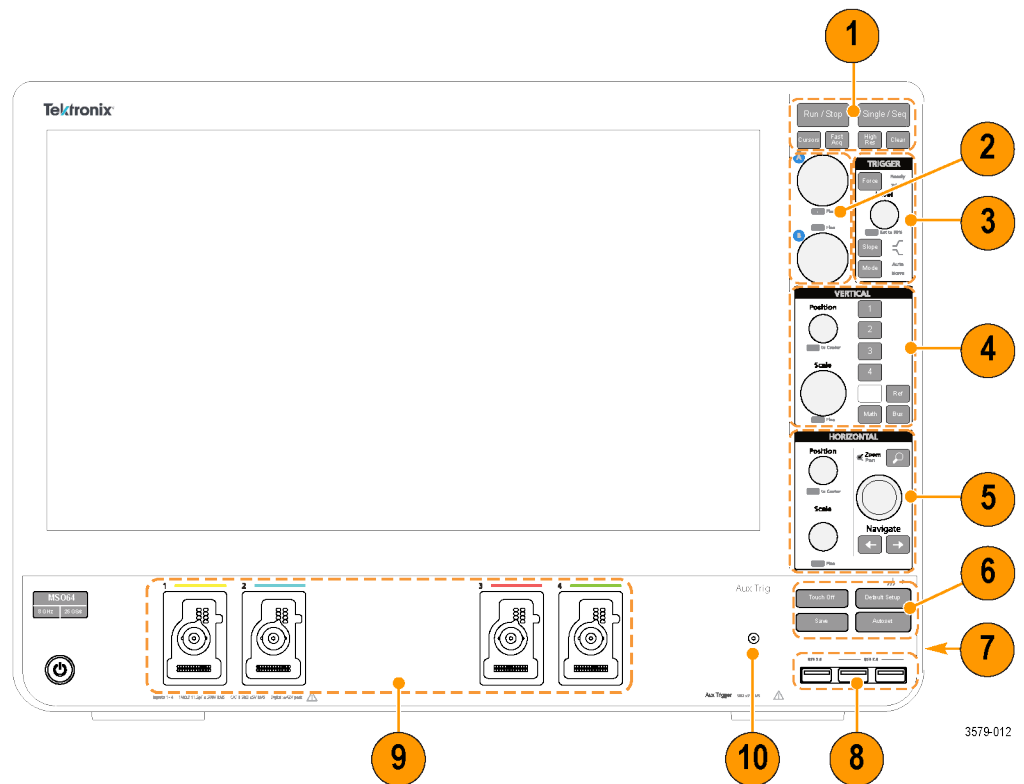


Figure 8: 6 Series MSO controls

1. Acquisition and Cursors controls:



3514-011

- **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 130.
- **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

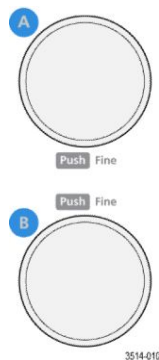
transient events such as glitches and runt pulses. It is helpful in finding elusive signal anomalies. Fast acquisition mode can also display waveform phenomena at an intensity that reflects their rate of occurrence.

- **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.

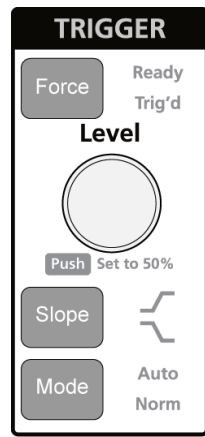
## 2. Multipurpose knobs:



- **Multipurpose knobs (A, B)** The Multipurpose knobs A and B move cursors and set parameter values in configuration menu input fields. Selecting an menu field that can use a Multipurpose knob assigns the indicated knob to change the value in that input field. 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:

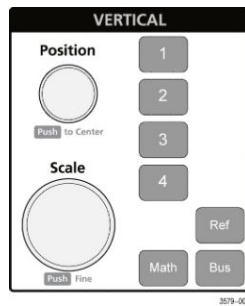


3514-012

- **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 for a trigger (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.



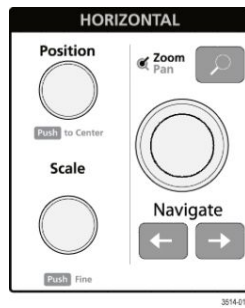
#### 4. Vertical controls:



- **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 turn on (display), select, or turn 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.

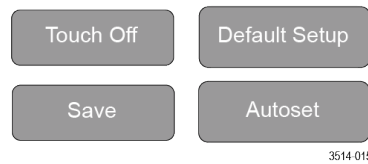
## 5. Horizontal controls:



- **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 74.
- **Zoom** knob (center 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. Press and hold a front panel navigate button to continue moving to the next search point in that direction. See [Badges](#) on page 63.

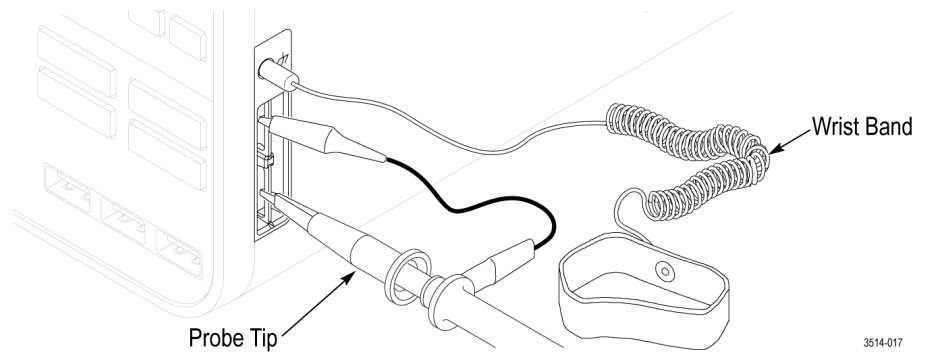
The front panel **Navigate** buttons can also be used for the **Previous** and **Next** button functions on measurement badges.

## 6. Miscellaneous controls:



- **Touch Off** turns touch screen capability off. The **Touch Off** button is lighted when the touch screen is turned off.
- **Save** is a one-push save operation that uses the current **File > Save As** settings to save screen shots (including open menus and dialog boxes), waveform files, instrument settings, and so on, as follows:
  - If a **File > Save** or **File > Save As** operation has occurred since the last instrument startup, pushing **Save** 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 **Save** opens the **Save As** configuration menu. Select a tab to select the 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 **Save**, the same type 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 100.

## 7. Ground and Probe Compensation connectors:



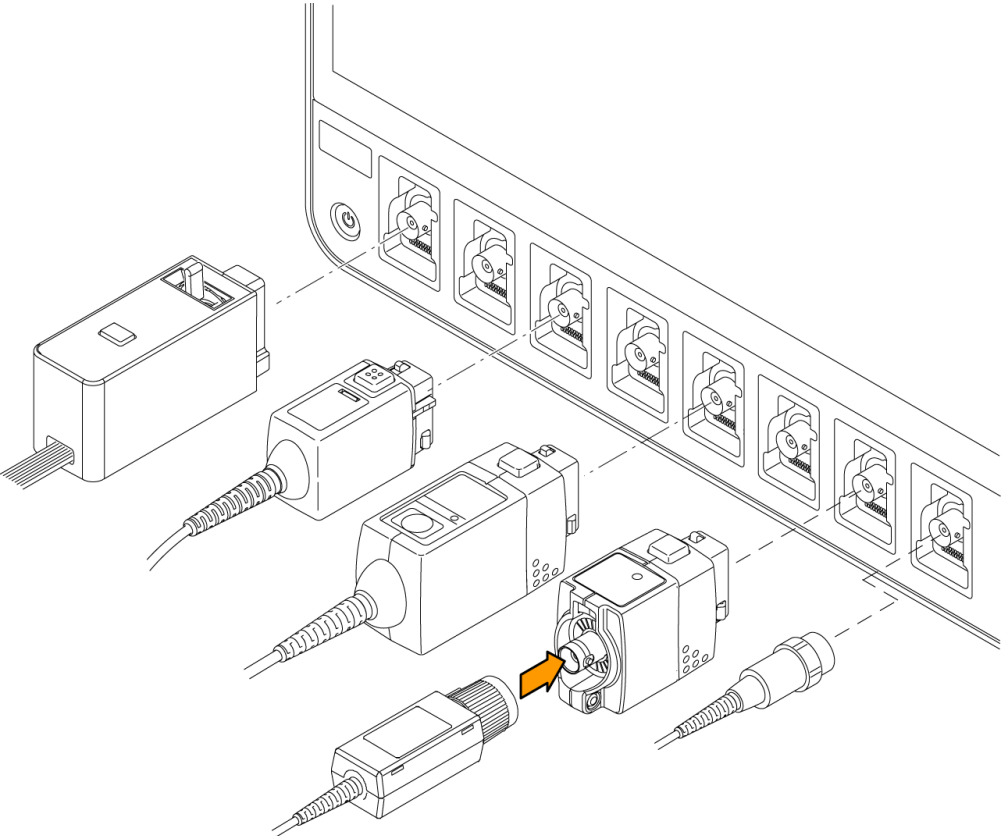
- The Ground and Probe Compensation connectors are located at the lower right side of the instrument, near the front panel. The Ground connector (the small hole in the case) provides an electrically grounded (through a resistor) connection point to attach an anti-static wrist strap, to reduce electrostatic 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 83.

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

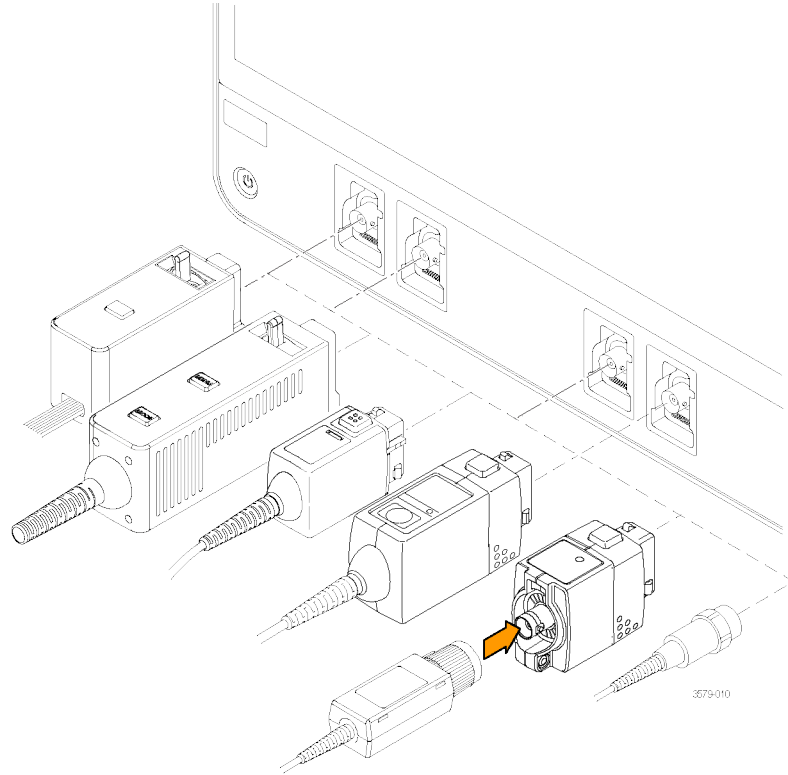


- USB ports are located at the lower right corner of the front panel, and on the rear panel. Connect USB flash drives to which you can save or recall data (such as instrument software updates, waveforms, settings, and screen captures), or connect peripheral devices such as a mouse or keyboard.

9. FlexChannel probe connectors:



3514-019



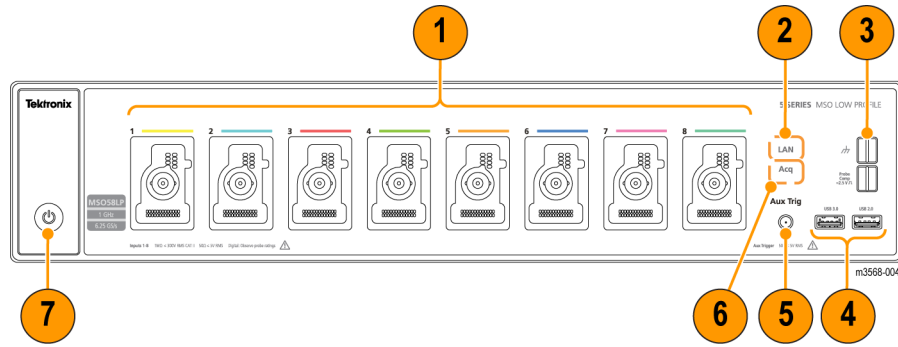
- FlexChannel connectors support all TekVPI+ and TekVPI measurement probes, BNC passive probes, the TPL058 FlexChannel Logic Probe, and BNC cables. You connect most probes simply by pushing them into the connector until the probe seats with a click. See [Connecting Probes](#) on page 40.

#### 10. Aux Trig trigger input connector:

An SMA connector to which you can connect an external trigger input signal. Use the AUX In trigger signal with the Edge trigger mode.

## MSO58LP front panel controls and connections

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



### 1. FlexChannel<sup>®</sup> probe connectors:

FlexChannel<sup>®</sup> connectors support all TekVPI+ and TekVPI measurement probes, BNC passive probes, the TLP058 FlexChannel<sup>®</sup> 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 40

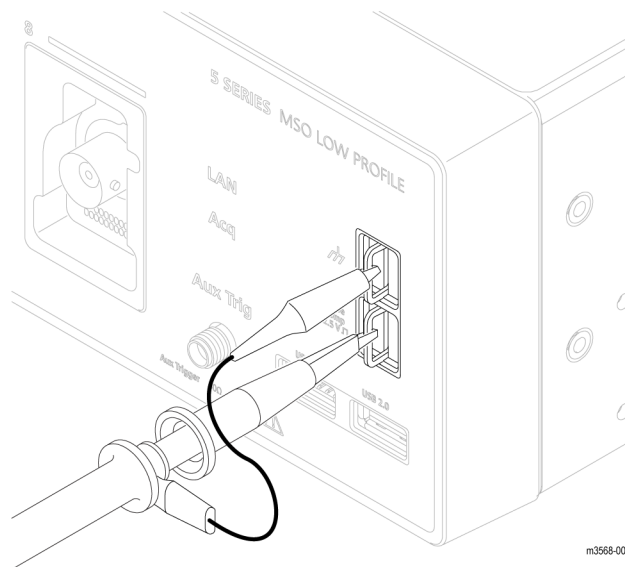
### 2. LAN Status LED:

Shows the network connection and activity status:

- Off - No power to instrument
- Green - Network connection is good
- Red - Network connection has fault or is 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 83.

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 connector to which you can connect an external trigger input signal. Use the AUX In trigger signal with the Edge trigger mode.

### 6. Acq status LED:

Shows the instrument trigger/acquisition status:

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

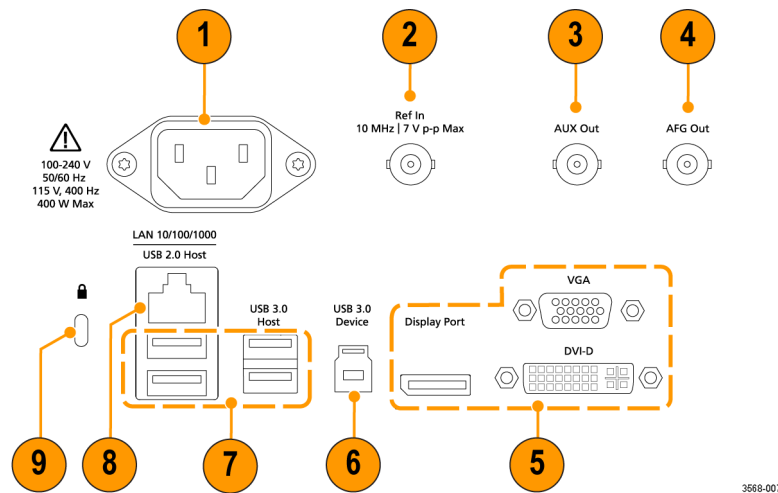
### 7. Power On/Standby button:

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.

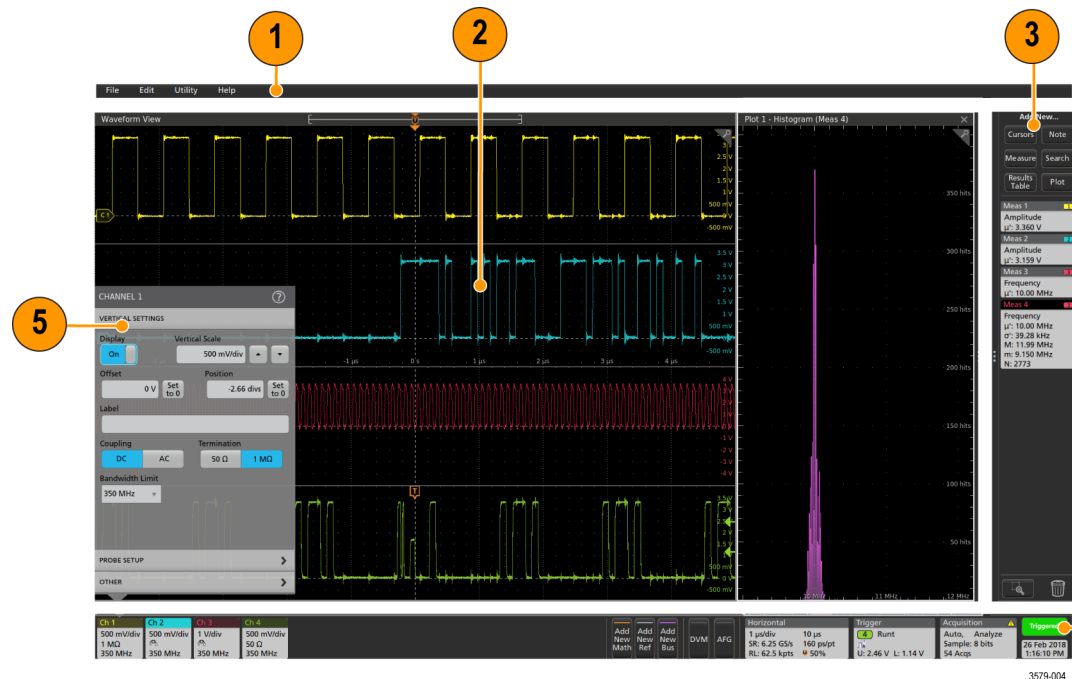


1. **Power cord** connector. Use only the power cord specified for this product and certified for the country of use.
2. **Ref In** lets you connect a high-precision 10 MHz reference signal to the oscilloscope for more accurate measurements.
3. **AUX Out** generates a signal transition on a trigger event, outputs a 10 MHz reference signal, or outputs a synchronization signal from the AFG.
4. **AFG Out** is the signal output for the optional Arbitrary Function Generator (AFG) feature.
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.



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 59.

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.

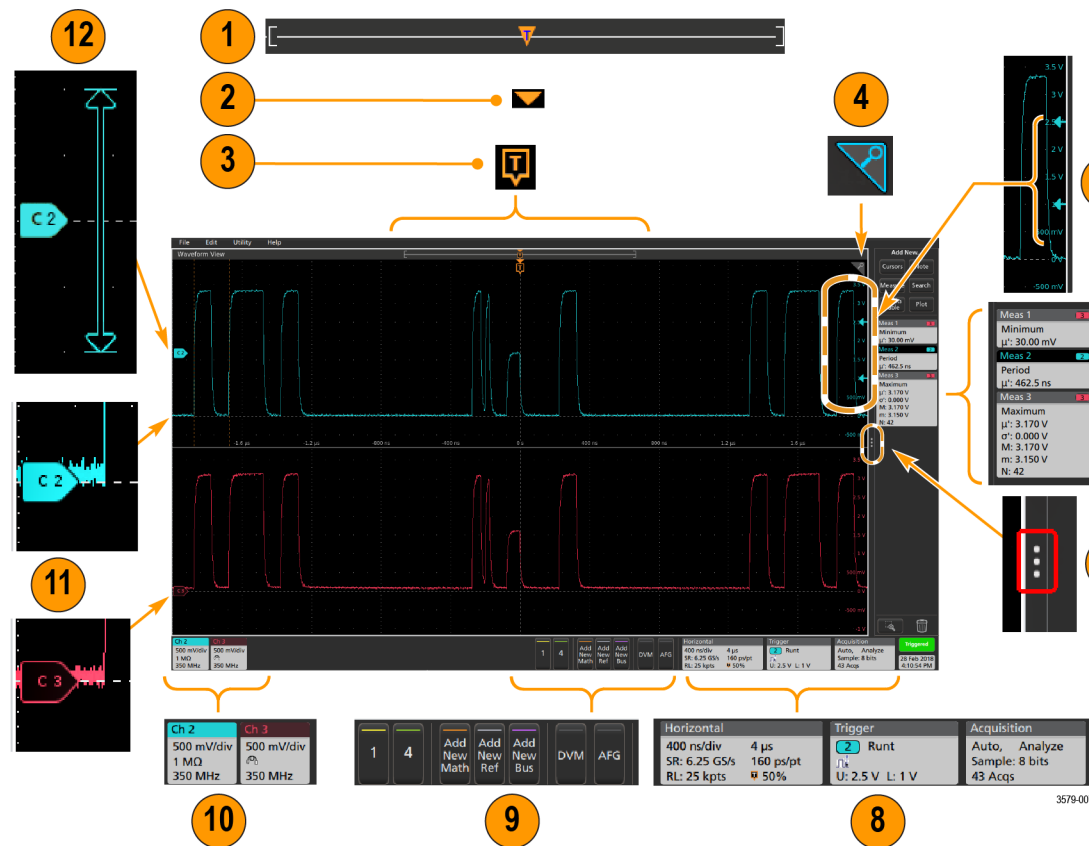
3. The **Results Bar** contains controls for displaying cursors, adding notes, plots, and result tables to the screen, and add measurements to the Results bar. The controls are:
  - The **Cursors** button displays on-screen cursors in the selected view. Touch and drag, or use the Multipurpose knobs, 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 **Results Bar**. See [Badges](#) on page 63. See [Add a measurement](#) on page 112. See [Add a Search](#) on page 127.
  - The **Draw-a-Box** button at the bottom of the **Results Bar** lets you toggle between drawing a box on the screen to zoom in on an area of interest, or drawing areas to define visual trigger conditions.
  - The **Trash Can** icon lets you drag Channel, Waveform, Measurement, and Search badges to the **Trash Can** to delete them.
4. The **Settings Bar** contains 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 a badge to open its configuration menu. See *Badges* on page 63.

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 72.

## 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.



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



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



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



When in Zoom mode, the Waveform Record View is replaced with the Zoom Overview. See [The Zoom user interface elements](#) on page 74.

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 63. See [Add a measurement](#) on page 112.
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 63.
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 Math**, **Add New Ref**, and **Add New Bus** buttons add the corresponding signal 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. Double-tap a badge to open its associated configuration menu. See [Badges](#) on page 63. See [Configuration menus](#) on page 72.

If you add more Channel or Waveform badges than can fit in the waveform badge display area, tap the scroll buttons at each end of the waveform badge area to scroll and display hidden badges.

11. The Waveform Handles on each waveform identify the source of that waveform (Cx for channels, Mx for Math waveforms, Rx for Reference waveforms, Bx for bus waveforms). The waveform handles are at the zero-volt level of the waveform by default. 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, each individual digital signal labeled D0–D7 and displayed with a different color.

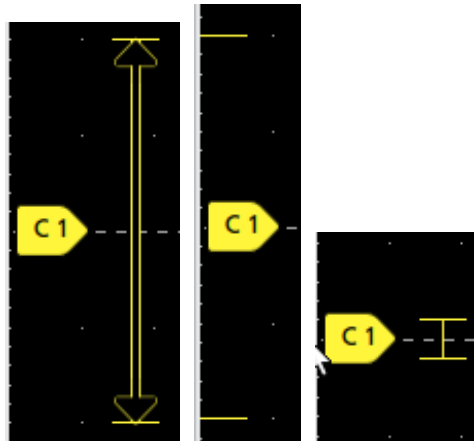


Double-tapping a digital waveform handle opens the digital channel configuration menu.

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

12. The probe Dynamic Range Limit Markers are displayed just within the left-hand graticule edge, based at the channel vertical trace handle position and extending up and down to the dynamic range limits of the probe. The markers are only displayed if compatible probes are used. Signals must be within the probe dynamic range for the oscilloscope to correctly display and measure the signals.

The markers are displayed, for about three seconds, after any **Offset**, **Position**, or **Scale** control change that leaves the channel dynamic range limits within the acquisition window. After about three seconds the markers become short lines at the left edge of the graticule. If the dynamic range is too small to display the arrows, the arrows are omitted. Examples of all three marker versions are shown.



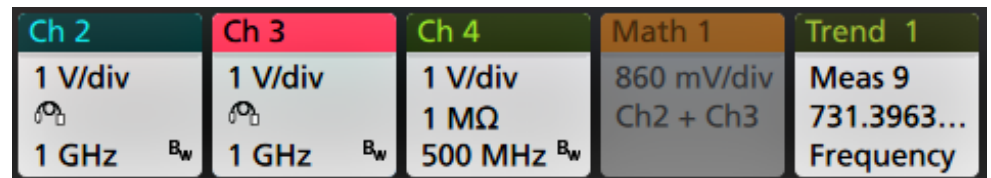


## Badges

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.

### Channel and Waveform badges

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



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



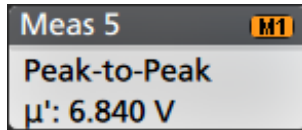
You can drag Channel and Waveform badges to change their position in the **Settings bar**, or drag them into the **Trash Can** icon to turn them off.

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

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

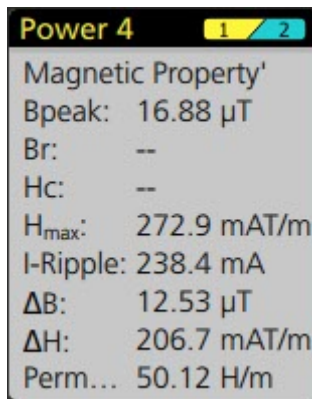
### Measurement badges

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

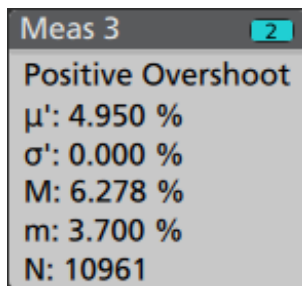


Double-tap a Measurement badge to open its configuration menu to change or refine settings. The default measurement badge readout shows the measurement's mean ( $\mu$ ) value.

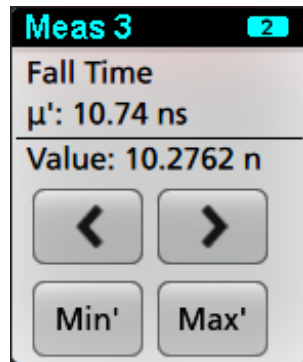
Some measurements and their badges are only available as options. For example, Power measurements are only listed in the Add New Measurement menu if the PWR option is installed.



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.

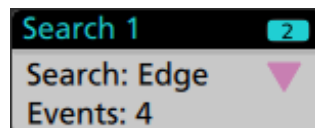
The prime symbol (') shown on measurement readings and Min/Max buttons indicates that the value shown (or moved to in the case of **Min/Max** buttons and waveforms) is from the current acquisition. Lack of a prime symbol means the value is from all acquisitions.

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 can drag Measurement badges to change their position in the **Results** bar, or drag them into the **Trash Can** icon to delete them.

### Search badges

**Search** badges are also shown in the Results Bar, below the Measurement badges. A search badge lists the search source, search type, and the number of search event occurrences in the current acquisition. The instrument 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.

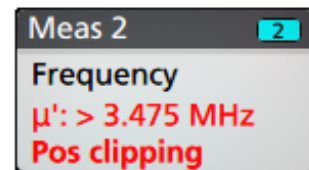
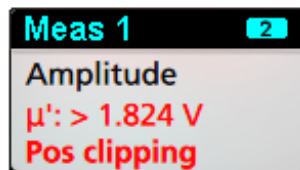
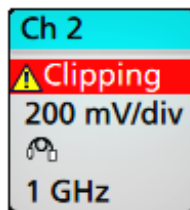
You can drag Search badges to change their position in the **Results** bar, or drag them into the **Trash Can** icon to delete them.

### Signal Clipping and Bad



**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.

## Error Messages and Badges

This instrument shows a warning triangle symbol and an error message abbreviation in a Channel badge when an error occurs.




To remove the message from the badge, clear the error as indicated in the table.

**Table 5: Probe errors**

Error message	Description
Prb Comm	Accessory communication timed out. Please re-attach the accessory.
Prb ROM	Unable to read probe ROM. Please re-attach the accessory.
Unsup	Accessory is unsupported.
Prb Fault	Critical accessory fault. Please re-attach the accessory. If the problem persists, contact Tektronix service.
Over Rng	The signal voltage or current is over range. Please reduce the signal amplitude.
Temp	The probe has experienced an over temperature condition. Please remove the probe from the high temperature area.
No Tip	No probe tip detected. Please install a compatible probe tip.
Tip Fault	The probe tip has a fault. Please remove and replace the probe tip.
S-param	Error during S-parameter transfer. Please reattach the probe. If the problem persists, contact Tektronix Service.

## System badges



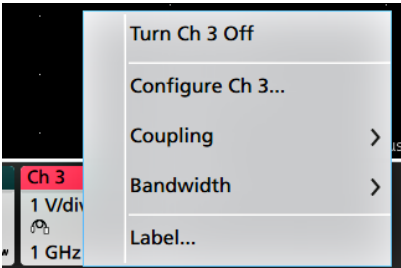
System badges (in the **Settings** bar) display the main Horizontal, Trigger, and Acquisition settings. You cannot delete System badges.

Horizontal	Trigger	Acquisition
1 $\mu$ s/div      10 $\mu$ s SR: 3.125 GS/s    320 ps/pt RL: 31.25 kpts    50%	2 Runt  U: 2.28    L: 800 m	Auto, Analyze High Res: 12 bits 10.379 kAcqs

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

Badge type	Selected	Unselected	Turned off or in use <sup>1</sup>
Channel or Waveform			
Measurement			N/A

## Moving waveform and measurement badges

You can move waveform and measurement badges within their display bars to meet your measurement needs. Simply touch and drag the badge to a new position.

### Moving waveform badges in the Settings Bar

- A waveform badge can only be moved within the Settings Bar.
- Dragging a waveform badge to a new location selects that waveform.
- Dragging a badge to a new location causes the non-selected badges to move slightly to create the position at which to insert the badge.
- To move a badge to a position that is off screen of the displayed bar badges (scroll buttons are present), drag the badge that you're moving onto one of the scroll buttons. The badges that are off screen shift onscreen one at a time until you move the badge off of the scroll button and position it in the displayed badges.
- Changing the order of waveform badges or badge groups also changes the order of displayed waveforms in the Waveform view. The order of badges or badge groups (left to right) in the Settings bar determines the order of slices in the display (top to bottom).
- Once you have moved any waveform badge (Channel, Math, Ref, Bus, Trend) on the Settings bar to a new position, adding a new badge adds that badge to the right of the existing badges. This differs from the default badge addition action, which listed badges together by category (Channels, Math, Ref, Bus) and arranged the badges in numerical order within each badge

<sup>1</sup> 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.

category. To restore the default badge addition method, tap **File > Default Setup** and add the waveform badges back onto the Settings bar.

- Changing the order of bus waveform badges changes the order of the tabs in a bus decode Results table.

### Moving badges in the measurement Results Bar

- A Measurement or Search badge can only be moved within the Results Bar.
- Dragging a badge to a new location causes the non-selected badges to move to create the position at which to insert the badge.
- Once you have moved any Results badge to a new position, adding a new Measurement or Search badge adds that badge to the bottom of the Results bar badges.
- To move a badge to a position that is off screen of the displayed bar badges (scroll buttons are present), drag the badge that you're moving onto one of the scroll buttons. The badges that are off screen shift onscreen one at a time until you move the badge off of the scroll button and position it in the displayed badges.
- Changing the order of Measurement badges in the Results bar changes the order of measurements shown in a Results table.
- Changing the order of Search badges in the Results Bar changes the order of the tabs shown in a Search Results table.
- Changing the order of harmonics badges in the Results bar changes the order of the tabs shown in a Harmonics Results table.

### Badge grouping

You can group signal badges in the Settings bar to display multiple waveforms in a single slice. See the *Group signal badges in the Settings bar* topic.

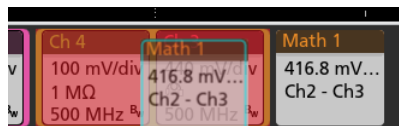


## Group signal badges in the Settings bar

You can group signal badges in any combination of channel, waveform, math, reference and bus badges to display related waveforms in the same display slice. Use this feature to group related signals for easier visual comparisons.



To group one or more signal badges, drag a badge onto another badge until the background badge turns red, then release. Repeat to add other badges to a group. Each group is shown in its own slice as a set of overlaid waveforms.



### Badge group characteristics

- Badges in a group are listed left to right in the order they were added to the group.
- The badge group color is the color of the last selected badge in the group.
- Use standard touch and menu interactions to select and change individual badge settings.
- You can drag and move badge groups the same way as individual badges.
- The order of badges or badge groups (left to right) determines the order of slices in the display (top to bottom).
- To ungroup a single badge, drag it vertically and release. You can also ungroup a single badge by touching and holding a badge in the group to open

its right-click menu, and select Ungroup Badge(x) to ungroup the selected badge.

- To ungroup the entire group, touch and hold any badge in the group to open its right-click menu. Select to ungroup the selected badge or ungroup all of the badges.

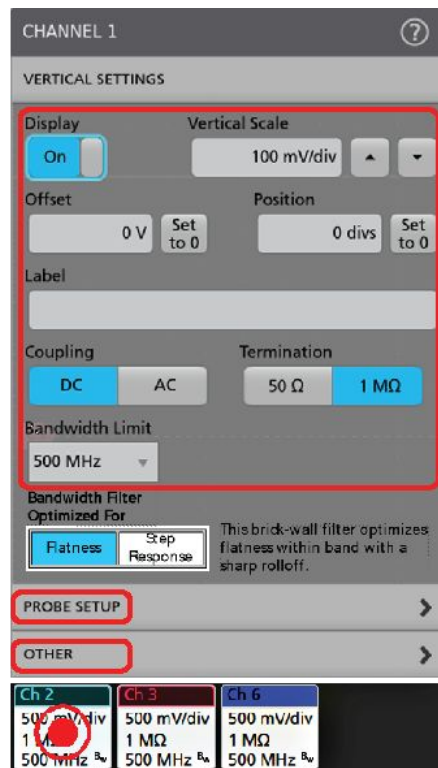
### Badge group caveats

- You cannot drag badges within a group to change their order.
- You cannot add one badge group to another.
- You cannot drag a badge group onto an individual badge to group those badges.
- You cannot drag a badge group onto the Trash icon to ungroup or delete those badges.

## 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.

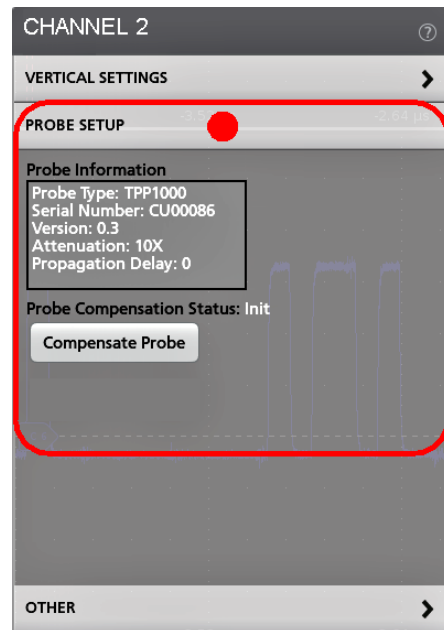
Double-tap an item (badge, **Waveform View** or **Plot View**, cursor readouts, note text, and so on) to open its configuration menu. For example, double-tap a Channel badge in the **Settings Bar** to open its configuration menu.



3579-011

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

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



Tap anywhere outside a configuration menu to close it.

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

## The Zoom user interface elements

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



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

**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 (see 5). 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 toggles between drawing a zoom box (default mode) and drawing areas for the **Visual Trigger** function. The button is located at the bottom of the **Results Bar**.

A zoom box lets you quickly draw a box around an area of interest in the Waveform or Zoom Overview. Drawing a box immediately puts the oscilloscope into zoom mode. To draw a zoom box, tap the Draw-a-Box button (while in Zoom mode), then touch and drag on the waveform to draw a box waveform. You can continue to draw zoom boxes until you single tap anywhere on the screen or open a menu.

To toggle between zoom mode and **Visual Trigger** mode, double-tap the Draw-a-Box button and select **Visual Trigger**. Search for the **Visual Trigger** topics in the oscilloscope embedded Help for more information on **Visual Trigger**.

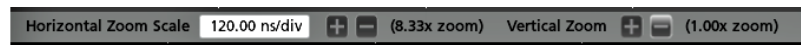
5. The **Zoom View** shows the zoomed waveforms, as marked by the Zoom Box, in the Zoom Waveform Record View. Use pinch and/or drag options in the zoom view to change the zoomed area of interest.

---

**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.

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

**Table 6: 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, <b>Add New Math</b> , <b>Add New Reference</b> , or <b>Add New Bus</b> button.	Click an inactive channel button, <b>Add New Math</b> , <b>Add New Reference</b> , or <b>Add New Bus</b> 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 <sup>2</sup> , search, horizontal).	Tap the badge.	Click the badge.

Task	Touchscreen UI action	Mouse action
Open a configuration menu on any item (all badges, views, cursor readouts, labels, and so on).	Double-tap the badge, view, or other object.	Double-click the badge, view, or other object.
Open a right-click menu (badges, views).	Touch and hold on the badge, Waveform View, Plot view, or other screen item until a menu opens.	Right-click the object.
Close a configuration menu <sup>3</sup> .	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 <sup>4</sup> .	Touch and hold on a note and quickly <sup>5</sup> 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. Touch and hold two fingertips on the waveform view, move them together or apart vertically or horizontally, lift from screen; repeat.	Left-click a channel, waveform, or <b>Horizontal</b> badge and click on the Scale buttons.
Increase or decrease the zoom area (while in Zoom mode)	Touch and hold two fingertips on the waveform view, move them together or apart vertically or horizontally, lift from screen; repeat.	Click the + or - buttons on the Zoom Title bar. Click the <b>Draw-a-Box</b> 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.

<sup>2</sup> Not all measurement or search badges display navigation buttons.

<sup>3</sup> Some dialog boxes will not close until you click an OK, Close, or other button in the dialog.

<sup>4</sup> Notes are screen objects and are not associated with any particular waveform channel or slice.

<sup>5</sup> Start to move the note as soon as selected (highlighted), otherwise the UI opens the right-click menu.

Task	Touchscreen UI action	Mouse action
Close or open the <b>Results Bar</b> to increase the <b>Waveform View</b> area.	Tap on the <b>Results Bar Handle</b> (three vertical dots in border) or anywhere in the border between the <b>Waveform View</b> and the <b>Results Bar</b> .	Click the <b>Results Bar Handle</b> (three vertical dots in border) or anywhere in the border between the <b>Waveform View</b> and the <b>Results Bar</b> . Click and drag the Results Bar divider.
Change the position of badges in the <b>Settings Bar</b> or <b>Results Bar</b> . See the Help topic <i>Rearranging waveform and measurement badges</i> for more information on moving badges.	Touch and drag the badge to a new position in the same bar.	Click and drag the badge to a new position in the same bar.

## Accessing application help

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

### 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

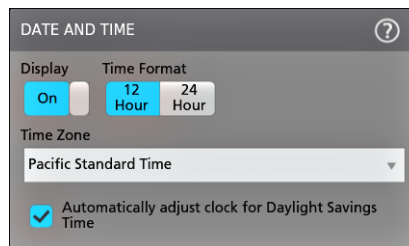
## Set the time zone and clock readout format

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 oscilloscope 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. 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**.

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

## Functional check

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

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 oscilloscope 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<sub>p-p</sub>).
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

Installing the latest firmware helps ensure that your oscilloscope is taking the most accurate measurements.

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. The installation process does not remove user-created files, but it is a good idea to back up important files before an update.

### Update oscilloscope firmware from a 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.tek.com/product-support**.
2. Enter the oscilloscope model number in the search field and click **Go**.
3. Scroll down the screen and click the **Software** tab.
4. If the listed available firmware version (Windows or non-Windows) 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:
  - a. Non-Windows oscilloscopes detect the USB drive with the firmware file and start the installation process. Follow on-screen instructions to install the firmware.
  - b. For Windows oscilloscopes, close the TekScope program before updating the firmware. Open the File Explorer and run the firmware update file from the USB drive, or copy the firmware update file to the desktop and run the file from there. Follow on-screen 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.

---

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

To confirm the firmware installation:

- a. Tap **Help** > **About** in the Menu bar.
- b. 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 (9 °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 oscilloscope 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 oscilloscope 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.

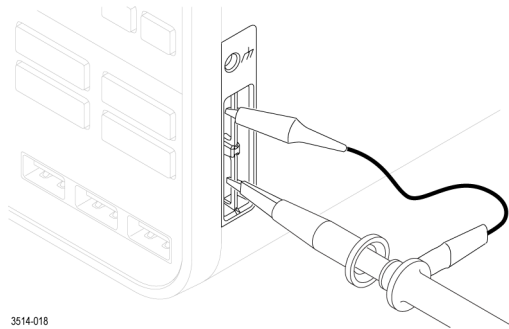
---

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

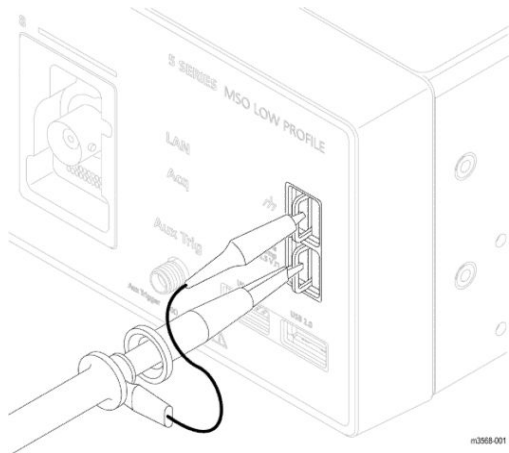
---

**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.
2. Connect the probe tip and ground lead of the probe to the PROBE COMP terminals on the lower right of the oscilloscope (see following image).



**Figure 9: Probe Comp connections on the 5 Series/6 Series MSO instruments.**



**Figure 10: Probe Comp connections on the MSO58LP.**

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. On the MSO58LP, select **File > Autoset** from the menu bar. The screen displays a square wave.
6. Double-tap the badge of the channel that 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 oscilloscope 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.

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, and the IP address field does not already show an address, tap **Auto** to obtain the IP address information from the network. DHCP mode is the default mode.
  - If your network is not DHCP-enabled, or you need a permanent (non-changing) IP address for this instrument, tap **Manual** and enter the IP address and other values provided by your IT or system administrator resource.
4. Tap **Test Connection** to verify that the network connection is working. The LAN Status icon turns green when the instrument successfully connects to your network. If you have problems connecting to your network, contact your system administration resource for help.

The front panel LAN status LED colors on the MSO58LP display the network status:

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



## Mount a network drive from a standard instrument

Use this procedure to mount (map) a network Linux mount point or Windows shared directory on a standard (non Windows OS) instrument.

Prerequisites:

The oscilloscope must be connected to a network that has access to the directories to mount or unmount. See the *Connect to a network (LAN)* topic in the application Help. .

To mount a Linux network drive on the oscilloscope, the network Linux mount point (drive, host) that you want to mount must be exported. If the mount point is not exported, work with your organization's IT resource to get that location exported and accessible to your network.

To mount a Windows network drive on the oscilloscope, the network Windows directory location that you want to mount must be set to shared. If the Windows directory is not shared, work with your organization's IT resource to get that location shared and accessible to your network.

To mount a network drive on your standard oscilloscope:

1. Tap **File > File Utilities**.
2. Tap **Mount** to open the **Mount Network Drive** menu.
3. Select the drive letter to assign to the network drive from the **Drive Letter** list.
4. Tap **Name** or **IP** to specify how you will enter the host name (server) of the network mount location or PC.
5. Enter the network host name or IP address of the Linux mount point or Windows PC or server in the **Server Name** or **Server IP Address** field.  
Example: ACME-PC0205
6. Enter the path to the location of the mount point or shared directory of the server in the **Path** field.
  - Linux example: /opt/testing/batch1 (Linux requires forward slashes in path definitions. Linux assumes that the path starts at the root directory.)
  - Windows example: \\Users\adamt\Desktop\testing\batch1 (Windows requires back slashes in path definitions. The double back slashes (\\) denotes the root directory as the starting point for locating the specified directory.)
7. If access to this network location is controlled, enter the required information in the **User Name** and **Password** fields.
8. Tap **OK**. The oscilloscope mounts the drive and adds the specified drive letter to the **Drive** column in the **File Utilities** menu.

The oscilloscope displays an error message if it cannot mount the drive. Work with your organization's IT resource to verify that the access information is correct and to resolve the network access problems.

## Mount a network drive from a Windows 10 instrument

Use this procedure to mount (map) a network Linux mount point or Windows shared directory on a Windows OS instrument. Use the network drive to store or access files from the oscilloscope.

Prerequisites:

The oscilloscope must be connected to a network that has access to the directories to mount or unmount. See the *Connect to a network (LAN)* topic in the application Help. .

The oscilloscope must be added to your organization's network domain to mount network drives and be set to enable network discovery. Work with your organization's IT resource to configure the instrument.

To mount a Linux network drive on the oscilloscope, the network Linux mount point (drive, host) that you want to mount must be exported. If the mount point is not exported, work with your organization's IT resource to get that location exported and accessible to your network.

To mount a Windows network drive/folder on the oscilloscope, the network Windows folder location that you want to mount must be set to shared. If the Windows folder is not shared, work with your organization's IT resource to get that folder shared and accessible to your network.

To mount a network folder on your Windows OS oscilloscope:

1. Tap **File > File Utilities** to open the standard Windows Explorer window on the current directory.
2. Tap **This PC** in the path field to move to the This PC directory.
3. Tap and hold the **This PC** text on the left side of the pane to open the context menu.
4. Tap **Map Network Drive** to open the **Map Network Drive** menu.
5. Use standard Windows practices to set the drive letter and specify the path to the network folder. If access to the specified folder has security settings, you must enter the correct user and password information when requested.
6. The network folder is added to **This PC > Devices and Drives** listing.

If you have problems accessing a network folder, work with your organization's IT resource to verify that the folder is shared and that you are using the correct user name and password.

## Unmount a network drive from a standard instrument

Use this procedure to unmount (remove, disconnect) a network Linux mount point from a standard (non-Windows 10) instrument.

To unmount a network drive from your instrument:

1. Tap **File > File Utilities**.
2. Select the drive to unmount in the **Drive** column.
3. Tap **Unmount**. The instrument immediately unmounts the drive and removes it from the Drive column.

## Unmount a network drive from a Windows 10 instrument

Use this procedure to unmount (remove, disconnect) a network drive from a Windows 10 instrument.

To unmount a network drive from your Windows 10 instrument:

1. Tap **File > File Utilities** to open the standard Windows Explorer window on the current directory.
2. Tap **This PC** in the list to move to the This PC directory.
3. Select the network drive/folder to unmount.
4. Tap and hold, or right/click, the selected drive to open a context menu.
5. Tap **Disconnect** to unmount the drive. the drive is immediately removed from the list.

## 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 access your network-connected standard instrument (not running Windows 10) from a Web browser to display the instrument user interface on a PC.

This procedure describes how to remotely access the UI controls and screen for standard (non Windows 10) instruments. To remotely access the UI controls and screen for Windows 10 instruments, see

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 86
  - The IP address of the oscilloscope that you want to access. To determine the oscilloscope's IP address, select **Utility > IO** in the oscilloscope menu bar and view the network settings in the **LAN** panel.
  - You are accessing a standard (instrument that does not have the Windows OS option installed).
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<sup>®</sup>)**. 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.

---

**NOTE.** When you access the instrument from an e\*Scope browser, you cannot directly paste text (such as path, IP address information, and so on) from the PC to an instrument menu field. You must use an intermediate clipboard function that is available in the e\*Scope application.

Use the following steps to copy text from an e\*Scope-connected PC to the instrument:

- a. Open a connection to the instrument using **e\*Scope**.
- b. Select and copy the text on your PC.
- c. In e\*Scope, press **Ctrl-Alt-Shift** to open the **Clipboard** menu.
- d. Paste the text into the Clipboard field.
- e. Press **Ctrl-Alt-Shift** to close the browser Clipboard menu
- f. Use e\*Scope to open the instrument menu to which to paste content, and position the cursor in the field where you want to paste the text.
- g. Press **Ctrl-V** (on real keyboard or from virtual keyboard) to paste the text from the e\*Scope browser clipboard to the menu field.
- h. Repeat steps 4.b through 4.g to copy and paste other text from the PC to the instrument.

---

**NOTE.** When you access the instrument from an e\*Scope browser, you cannot directly paste text (such as path, IP address information, and so on) from the PC to an instrument menu field. You must use an intermediate clipboard function that is available in the e\*Scope application.

Use the following steps to copy text from an e\*Scope-connected PC to the instrument:

- a. Open a connection to the instrument using **e\*Scope**.
  - b. Select and copy the text on your PC.
  - c. In e\*Scope, press **Ctrl-Alt-Shift** to open the **Clipboard** menu.
  - d. Paste the text into the Clipboard field.
  - e. Press **Ctrl-Alt-Shift** to close the browser Clipboard menu
  - f. Use e\*Scope to open the instrument menu to which to paste content, and position the cursor in the field where you want to paste the text.
  - g. Press **Ctrl-V** (on real keyboard or from virtual keyboard) to paste the text from the e\*Scope browser clipboard to the menu field.
  - h. Repeat steps 2 through 7 to copy and paste other text from the PC to the instrument.
-

## Remote access to a Windows 10 instrument

Access to the graphical interface of a Windows 10 instrument requires installing software called TightVNC on both the PC and the oscilloscope. This procedure describes how to use TightVNC to access the instrument UI, and has links to the procedures to install TightVNC on the PC and the instrument.

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 86
  - The IP address or hostname of the oscilloscope that you want to access. Select **Utility > IO** in the oscilloscope menu bar and view the network settings in the **LAN** panel.
  - TightVNC software must be installed on both the Windows 10 oscilloscope and the PC used to remotely access the oscilloscope. See [Install TightVNC on the Windows 10 instrument](#) on page 92 and [Install TightVNC on a PC](#) on page 93.
1. On the PC, run the **TightVNC Viewer** application. If it is not on the Task Bar or the Startup menu, use the search window in the **Start** menu to locate the **TightVNC Viewer** application.

---

**NOTE.** Make sure that you select the **TightVNC Viewer** application. Do not select the **TightVNC Server** application.

---

2. Enter either the oscilloscope IP address or hostname in the **New TightVNC Connection** dialog and click **Connect**.
3. If an access password was set in the oscilloscope TightVNC Server, enter that password and click **OK**. The **TightVNC Viewer** opens and displays the oscilloscope screen.
4. To close the TightVNC viewer, click the **X** button in the upper right corner of the application.

### Install TightVNC on the Windows 10 instrument

The **TightVNC Server** application enables a PC to connect to and display the Windows 10 oscilloscope UI on the PC. You must install the TightVNC software on both the PC and the Windows 10 oscilloscope. This procedure describes how to install the software on the Windows 10 oscilloscope.

1. Download the **TightVNC** software:
  - a. Open a web browser and navigate to [www.tightvnc.com](http://www.tightvnc.com).
  - b. Under the **Download TightVNC for Windows (Version 2.8.8)** heading, click **Installer for Windows (64 bit)**.
  - c. 1. Click **Save File** in the dialog box to save the .MSI file to your PC.
  - d. Close the browser.
2. Navigate to where you saved the **TightVNC** installation file.
3. Copy the .MSI file to a USB memory device.

4. Attach the USB memory device to the Windows 10 oscilloscope.
5. Minimize the oscilloscope screen to display the desktop.
6. Use the Windows Explorer to navigate to and select the .MSI installation file on the USB device.
7. Right-click on the installation file and select **Install**.
8. Follow the on-screen instructions. Select **Typical** for the installation, leave all other settings at their default settings. Click **Next** when prompted.
9. Click **Install**.
10. Click **Yes** to allow software changes to the PC.
11. For best security, create user and administrator passwords for accessing the **TightVNC Server** from the **TightVNC Viewer**. The User password will be required whenever logging into the oscilloscope with the **TightVNC Viewer**.
12. Click **Finish**. The oscilloscope is now ready to enable connection from the TightVNC Viewer. See [Remote access to a Windows 10 instrument](#) on page 92.

## Install TightVNC on a PC

The **TightVNC Viewer** application connects the PC to the Windows 10 oscilloscope to display the oscilloscope UI on the PC. You must install the TightVNC software on both the PC and the Windows 10 oscilloscope. This procedure describes how to install the software on the PC.

1. Download the **TightVNC** software:
  - a. Open a web browser and navigate to [www.tightvnc.com](http://www.tightvnc.com).
  - b. Under the **Download TightVNC for Windows (Version 2.8.8)** heading, click **Installer for Windows (64 bit)**.
  - c. 1. Click **Save File** in the dialog box to save the .MSI file to your PC.
  - d. Close the browser.
2. Navigate to where you saved the **TightVNC** installation file.
3. Right-click on the installation file and select **Install**.
4. Follow the on-screen instructions. Select **Typical** for the installation, leave all other settings at their default settings. Click **Next** when prompted.
5. Click **Install**.
6. Click **Yes** to allow software changes to the PC.
7. Select **Do Not Change** for both password fields in the **Set Passwords** dialog box.
8. Click **Finish**. The PC is now ready to connect to a Windows 10 oscilloscope that has **TightVNC** software installed on it. See [Remote access to a Windows 10 instrument](#) on page 92.

## 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 oscilloscope 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 up to four probe tips and ground leads to the Probe Compensation connector (maximum of four channels at a time).
3. Turn on (display on screen) the connected 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.
12. To deskew additional channels:
  - a. Disconnect all probe tips except the reference probe from the Probe Compensation connections.
  - b. Connect up to three probe tips and ground leads to the Probe Compensation connector (maximum of four channels at a time).



- c. Repeat steps 3 through 12 for the additional channels 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 oscilloscope 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 to the oscilloscope.
2. Connect up to four probe tips and ground leads to the Probe Compensation connector (maximum of four channels at a time).
3. Turn on all the channels of the first four channels that you want to deskew.
4. Push the **Autoset** button.
5. Change the vertical Scale of all active channels to **500 mV/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 in the first set of four.
13. To deskew additional channels:
  - a. Disconnect all probe tips except the reference probe from the Probe Compensation connections.
  - b. Connect up to three probe tips and ground leads to the Probe Compensation connector (maximum of four channels at a time).
  - c. Repeat steps 3 through 13 for the additional channels 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 adapters. The instrument provides a ground connection to which to attach a wrist strap (on the Probe Comp ground connector).
- A cable that is left unconnected on a bench can develop a large static charge. Discharge the static voltage from all cables before connecting them to the instrument or device under test by momentarily grounding the center conductor of the cable, or by connecting a 50  $\Omega$  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.*

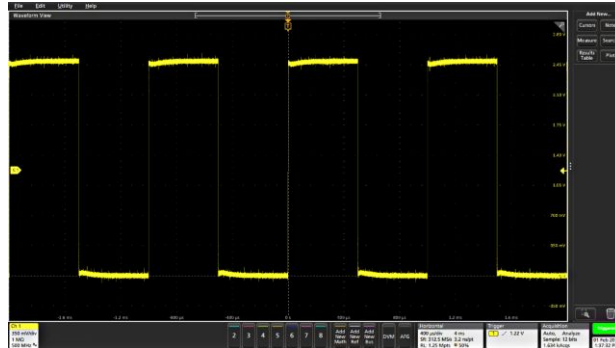
---

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 an available channel. The signal can be analog or digital.
2. Double-tap the **Trigger** badge and set the trigger source to that of the signal of interest.
3. Connect any other associated signal(s) to available channel input(s).
4. Add the channel waveforms to the Waveform view. See [Add a channel waveform to the display](#) on page 105.
5. Tap **File > Autoset** or push the front-panel **Autoset** button. When using the Stacked Display mode, the instrument analyzes the signal characteristics of the trigger source channel (analog or digital) and adjusts the horizontal, vertical, and trigger settings accordingly to display a triggered waveform for that channel. The Vertical scale is adjusted in each waveform slice of all active waveforms to maximize ADC utilization.



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

---

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

---

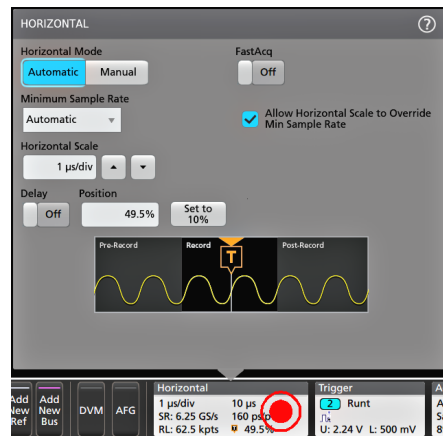
Autoset guidelines:

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

## Set Horizontal parameters

Use this procedure to set the horizontal time base parameters such as mode, 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.

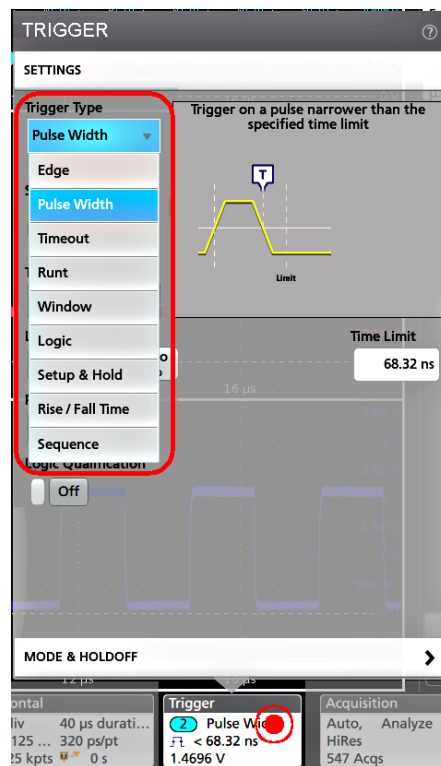


2. Use the menu selections to set horizontal parameters.
3. Tap the Help icon on the menu title for more information on these settings.

## 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.** For the MSO58LP, the AUX In external trigger input only supports Edge triggering.

---



---

**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 111

---



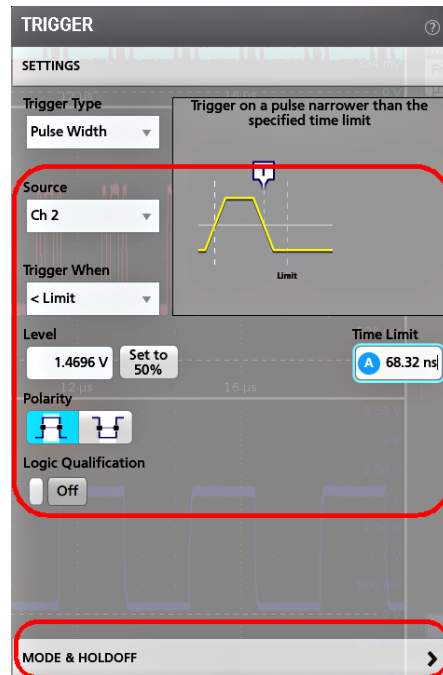
---

**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.

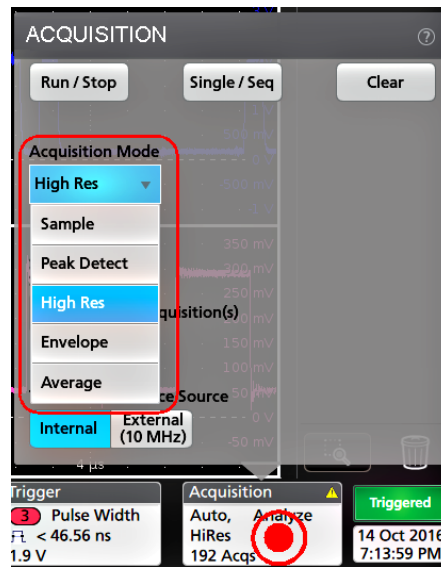


4. Tap the Help icon on the menu title for more information on these settings.
5. Tap outside the menu to close the menu.

## 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.

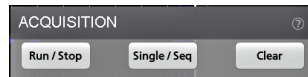


3. Tap the Help icon on the menu title for more information on these settings.
4. Tap outside the menu to close the menu.

## 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.

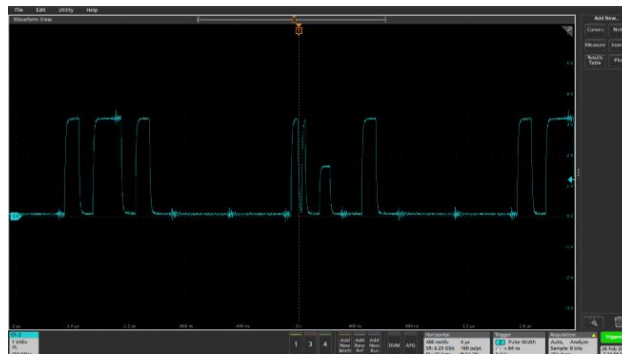
## Add a channel waveform to the display

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

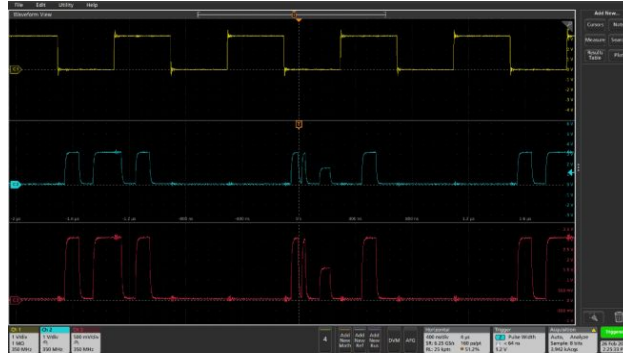
1. Connect signal(s) to the channel input(s).
2. Tap an Inactive Channel button (in the Settings bar) of a connected channel.



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

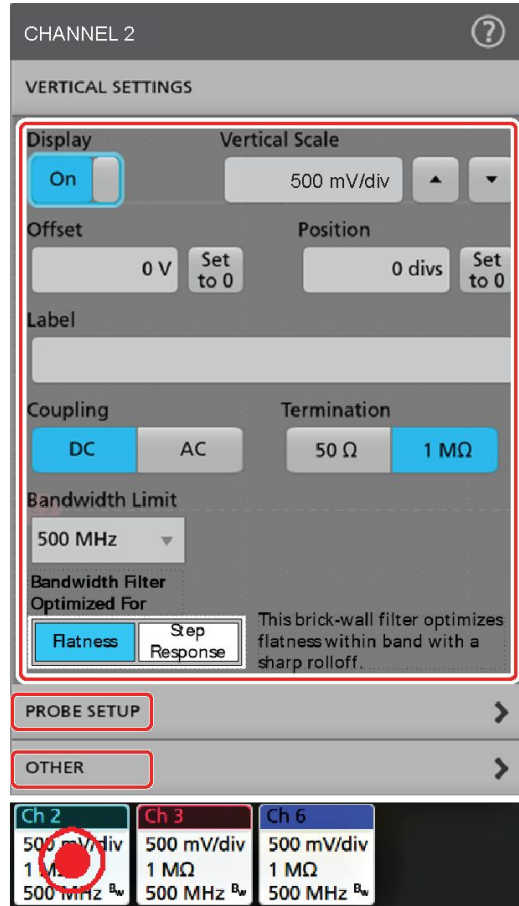
Use the channel and waveform configuration menus to set parameters such as vertical scale and offset, coupling, bandwidth, probe settings, deskew values, external attenuation values, and other settings.

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.

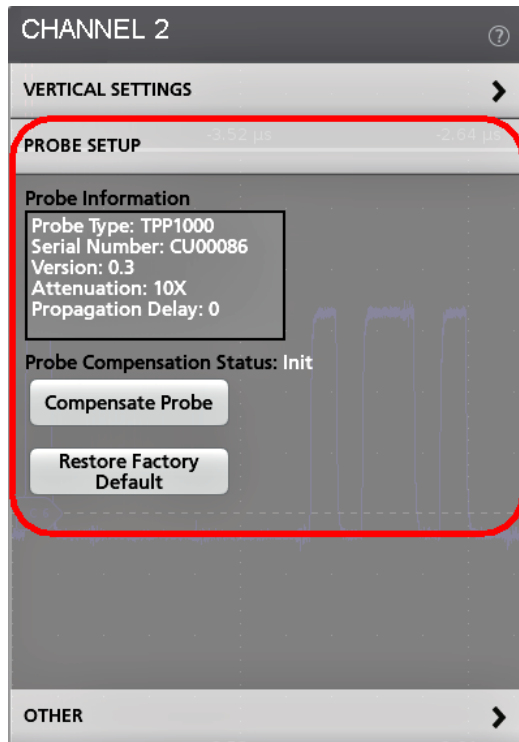


3514-021

Field or control	Description
<b>Display</b>	Enables <b>(On)</b> or disables <b>(Off)</b> displaying the channel waveform in the Waveform View window.
<b>Vertical Scale</b>	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
<b>Offset</b>	Sets the channel signal vertical offset.
<b>Set to 0</b>	Sets the channel signal vertical offset to 0 units.
<b>Position</b>	Double-tap the field to set the vertical position using the virtual keypad.
<b>Set to 0</b>	Tap to set the waveform zero volt level to the center of the slice or waveform view.
<b>Label</b>	Double-tap the field to add a label to the channel display using the virtual keypad.
<b>Coupling</b>	Tap to set the input coupling to DC or AC.
<b>Termination</b>	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.

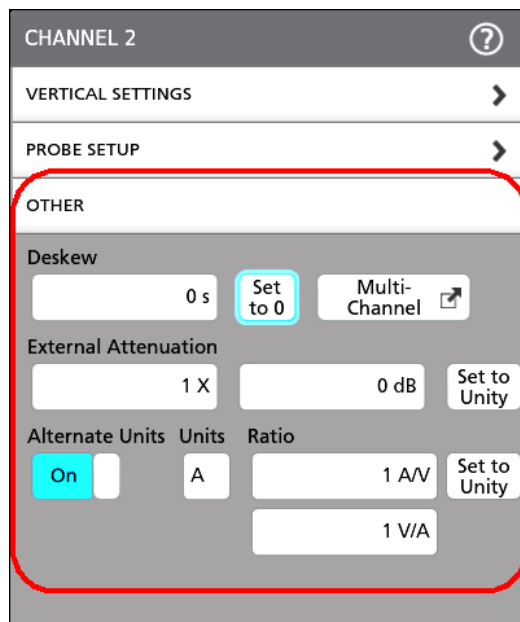
Field or control	Description
<b>Bandwidth Limit</b>	Tap to select the bandwidth limit from the drop-down list.
<b>Bandwidth Filter Optimized For</b>	<p>Tap to select a bandwidth filter that is optimized for flatness or step response. This selection is only available on 6 Series MSO instruments.</p> <p><b>Flatness</b> selects a brick-wall filter optimized for flatness within band with a sharp rolloff. Flatness filtering is not compatible with Peak Detect and Envelope acquisition modes.</p> <p><b>Step Response</b> selects a Bessel-Thompson filter that minimizes overshoot with a gradual rolloff.</p> <p><b>High Res</b> acquisition mode requires Flatness filtering.</p>

- 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
<b>Probe Information</b>	Displays the probe type, serial number, and attenuation, if it is available.
<b>Probe Compensation Status</b>	Displays the compensation status of the attached probe.
<b>Compensate Probe</b>	Starts the probe compensation procedure.
<b>Restore Factory Default</b>	Returns the compensation to the factory default settings.

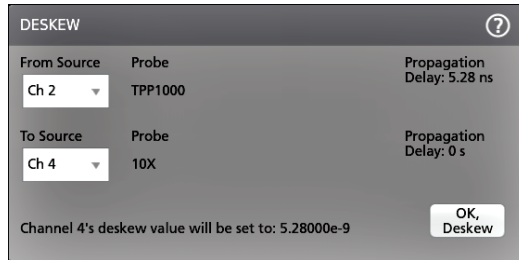
- To set up deskew, external attenuation, or alternate units, tap the **OTHER** setup panel.



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

Field or control	Description
<b>Ratio</b>	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.
<b>Set to Unity</b>	Sets the ratio to 1. This button is disabled when the Alternate Units switch is off.

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



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

- Tap outside the menu to close the menu.

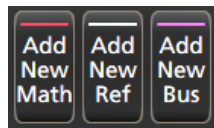


## Add a math, reference, or bus waveform

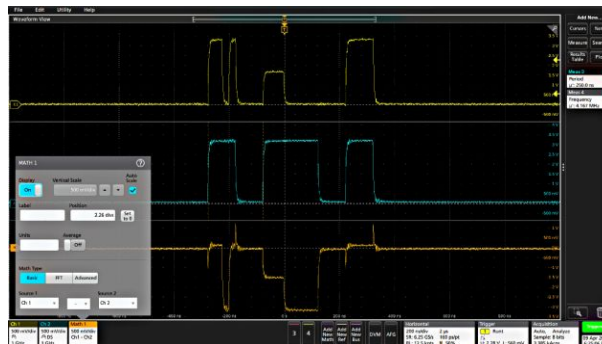
Math waveforms let you create new waveforms based on operations between two or more waveforms or by applying equations to waveform data. A reference waveform is a static waveform record displayed for comparison. Bus waveforms let you view and analyze serial or parallel data.

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.

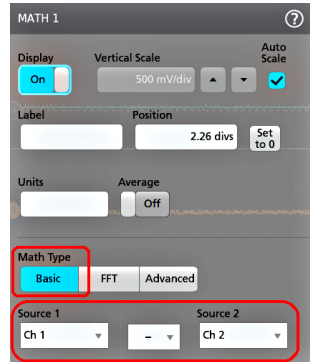


2. The instrument adds the waveform to the Waveform view, adds a Waveform badge to the Settings bar, and opens a configuration menu. This example shows adding a Math waveform.



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



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

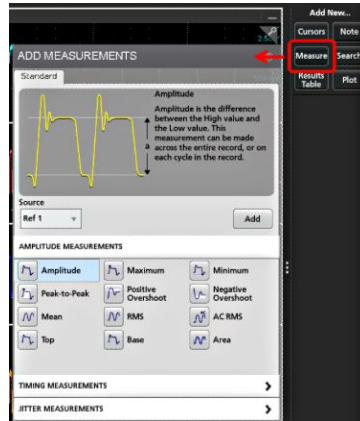
Use this procedure to select and add measurements.

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



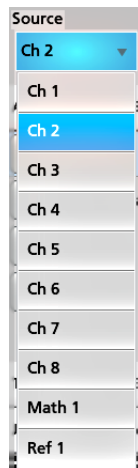
**NOTE.** Waveforms do not need to be displayed to be used for measurements, as long as the channel or waveform badge is on the **Settings** bar and is acquiring the signal to measure.

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



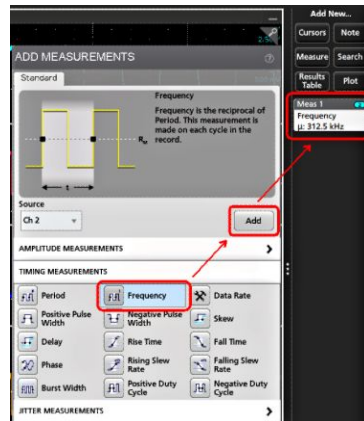
**NOTE.** If the menu shows tabs other than **Standard**, then optional measurement types have been installed on the instrument. Select a tab to show the measurements for that option.

3. Tap the **Source** field and select the measurement source. The list shows all available sources that are valid for the measurement.

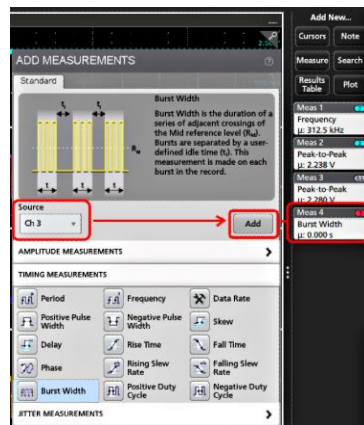


4. Select from the configuration menu panels, such as **Amplitude**, **Timing**, and **Jitter**, to display measurements for those categories.

5. Select a measurement and tap **Add** to add the measurement to the **Results** bar. You can also double-tap a measurement to add it to the **Results** bar.



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



8. Tap outside the **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 115.
10. Tap the Help icon on the menu title for more information on settings.

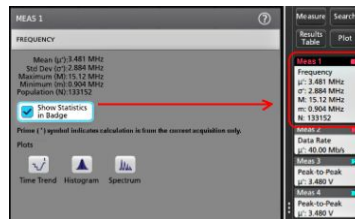
## Configure a measurement

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).

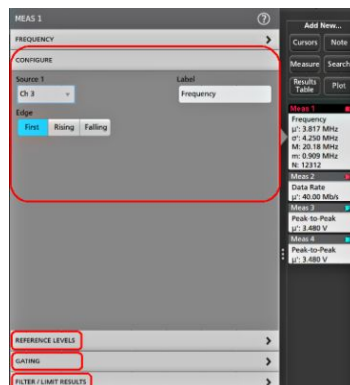
1. Double-tap a measurement badge to open its **Measurement** configuration menu.



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.

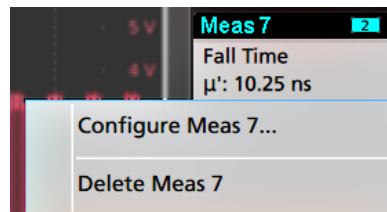


4. Use the available fields to refine the measurement conditions. Displayed fields depend on the measurement. Selection changes take effect immediately. Selection changes can also change fields in other panels.
5. Tap the Help button on the menu title for more information on this menu's settings.
6.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

---

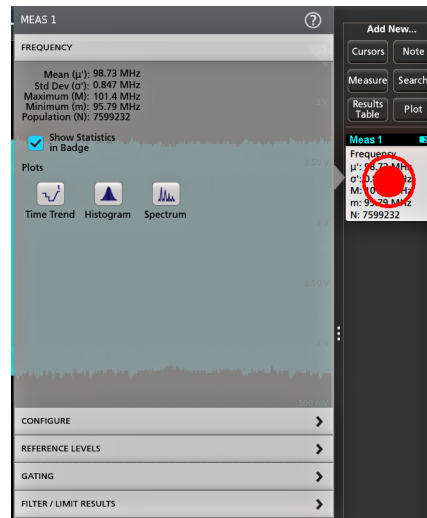
**NOTE.** *You can undo a measurement delete.*

---

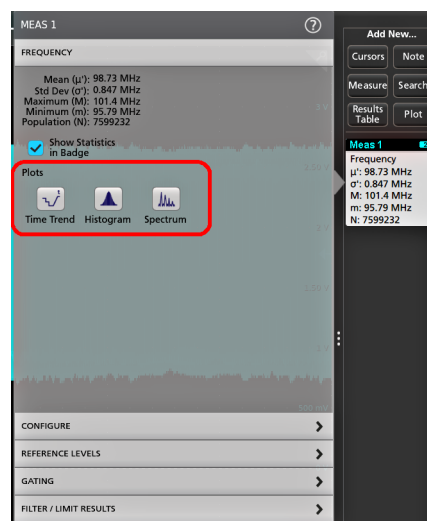
## Add a plot of a measurement

Measurement plots let you graph the distribution of waveform data point 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.

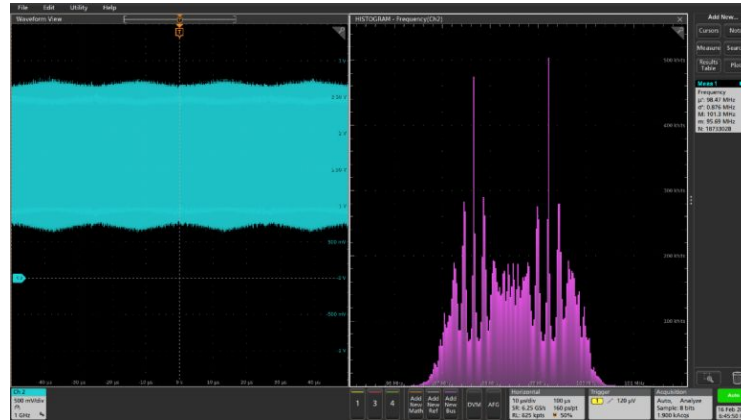
1. Double-tap a Measurement badge to open the **Meas** configuration menu.



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

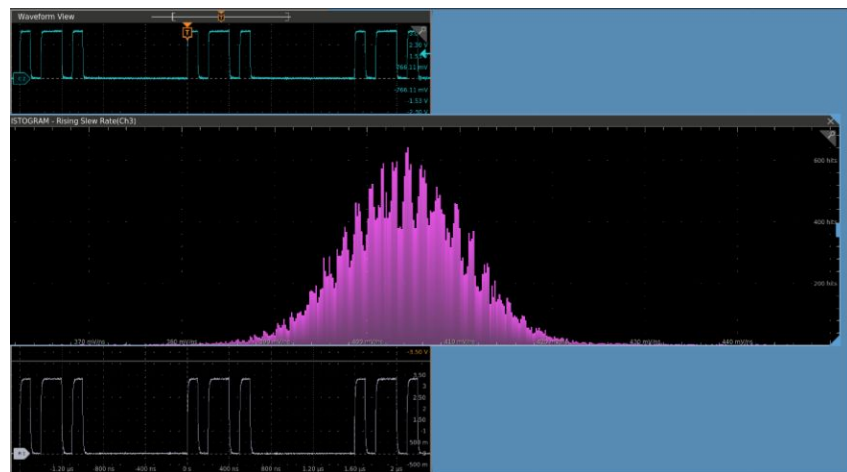


The following shows adding a Histogram plot.



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 Eye Diagram plot

Use this procedure to display an eye diagram of a waveform.

Prerequisite: An active signal for which you want to plot an eye diagram.

1. Tap **Add New... Plot** to open the Plot configuration menu.
2. Tap **Eye Diagram**.
3. Tap the **Source** drop down and select the signal source from the list.
4. Tap **Add**. The instrument adds the eye diagram in a separate plot window and adds a TIE measurement badge to the Results bar.
5. Double-tap in the Plot view to open a configuration menu for that plot.

## Add Mask Testing to an Eye Diagram plot

Use this procedure to add mask test segments to an eye diagram plot and display the mask segment hits information in the Results bar.

Prerequisites:

An eye diagram plot is on the screen.

A mask configuration file (.msk) that matches the time scale (unit interval) and vertical settings of the eye diagram.

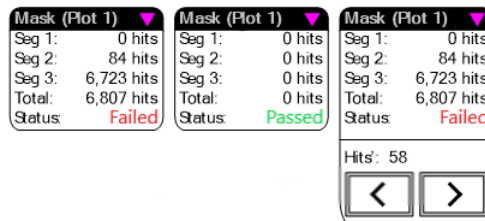
1. Double-tap in the eye diagram plot to open the **Eye Diagram** configuration menu.
2. Tap the **Mask** panel.
3. Use the **Mask** panel fields to select and load a mask configuration file and set the bit type for testing. See [Eye Diagram plot configuration menu](#) on page 444.

---

**NOTE.** The mask configuration file that you load must match the time scale (unit interval) and vertical settings of the active eye diagram.

---

This step also adds a **Mask** badge to the Results bar.



The **Mask** badge shows the number of segments in the mask, the hits per segment, and the pass/fail status. You can expand the Mask badge when acquisition is stopped to show the Navigation buttons.

Use the Navigation buttons to center the source waveform on the previous or next mask search mark where a mask hit occurred. In the case of mask hit groups (see the following text), the Navigation buttons center the source waveform on the beginning mask hit group search mark where the successive mask hits started.

The mask hit search marks on the source waveform are not precise due to the interpolated waveform used to create the eye diagram.

When a set of successive waveform sample points (for example, 50 sequential sample points) touch a mask, then the whole series of successive sample points is considered as one mask hit group. The successive mask hit groups are marked with a beginning and ending search mark, instead of individual search marks for each hit.

4. Tap the **Mask Test** button to set it to **On** and display the mask segments in the eye diagram.
5. Tap outside of the menu to close it.
6. To set the number of hits required to fail a mask test, double-tap the **Mask** badge in the Results bar to open the **Mask Hits** menu and set the hit fail value. See [Mask Badge configuration menu](#) on page 397.

## 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  $\frac{1}{2}$  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 450.

## Display a B-H curve plot for Magnetic Property measurement

Use this procedure to display a B-H curve plot for a power measurement. Requires option 5-PWR or 6-PWR.

To display a B-H Curve plot, you must be taking a Magnetic Property measurement (**Add New... Measurement > Power tab > Magnetic Analysis panel > Magnetic Property** measurement).

1. Double-tap the **Magnetic Property** measurement badge to open the configuration menu.
2. Tap **B-H Curve** to add a new B-H Curve Plot view to the screen.
3. Double-tap in the Plot view to open a configuration menu for that plot.

## Display an I vs (integral of) V plot for I vs V measurement

Use this procedure to display an I vs.  $\int V$  plot for I vs.  $\int V$  measurement. Requires option 5-PWR or 6-PWR.

To display an I vs.  $\int V$  plot, you must be taking a I vs.  $\int V$  measurement (**Add New... Measurement > Power tab > Magnetic Analysis > I vs.  $\int V$**  measurement).

1. Double-tap the **I vs.  $\int V$**  measurement badge to open the configuration menu.
2. Tap **I vs.  $\int V$**  to add a new I vs I vs.  $\int V$  Plot view to the screen.
3. Double-tap in the Plot view to open a configuration menu for that plot.

## Display an Inductance Curve plot for Inductance measurement

Use this procedure to display an inductance curve plot. Requires option 5-PWR or 6-PWR.

To display an inductance curve plot, you must be taking an Inductance measurement (**Add New... Measurement > Power tab > Magnetic Analysis panel > Inductance** measurement).

1. Double-tap an **Inductance** measurement badge.

The Measurement configuration menu is displayed.

2. Tap the **Inductance** button.

The Inductance Curve plot is displayed in a separate Plot view.

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

## Display a Safe Operating Area (SOA) measurement and plot

Use this procedure to display a SOA plot. This is an XY plot with a mask. Requires option 5-PWR or 6-PWR.

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

Use this procedure to display a trajectory plot for a power measurement. Requires option 5-PWR or 6-PWR.

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

Use this procedure to display a power/energy plot from the Power Quality measurement. Requires option 5-PWR or 6-PWR.

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 559.

## Display a Harmonics Bar Graph plot for the Power Harmonics measurement

Use this procedure to display a bar graph. Requires option 5-PWR or 6-PWR.

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.

## Display a Power Supply Rejection Ratio plot

Use this procedure to display a Power Supply Rejection Ratio plot for a power measurement. Requires option 5-PWR or 6-PWR.

To display a Power Supply Rejection Ratio 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 **Power Supply Rejection Ratio** panel.
5. Select the **Power Supply Rejection Ratio** measurement and tap **Add**.

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

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

## Display a Control Loop Response (Bode) plot

Use this procedure to display a Control Loop Response (Bode) plot for a power measurement. Requires option 5-PWR or 6-PWR.

To display an Control Loop Response 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 **Frequency Response Analysis** panel.
5. Select the **Control Loop Response** measurement and tap **Add**.

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

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

---

**NOTE.** *Gain Margin (GM) represents the margin value and not the actual value annotated in the Bode plot. The GM value indicates distance (basically how far) from the reference value (zero dB line). The margin is computed as zero minus the measured value. The measured value becomes negative when it is below 0 dB, then margin is positive and vice versa.*

---

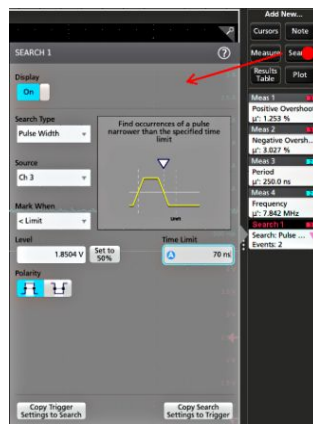


## Add a Search

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.

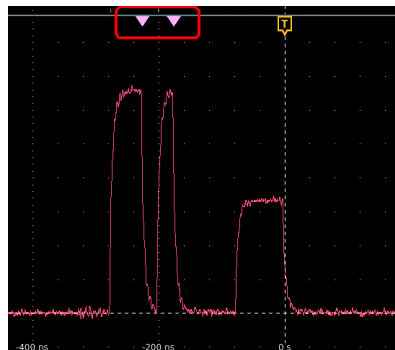
1. Display the channel or waveform signal on which to search. The waveform must be displayed to create a search for it.
2. Tap the **Add New...Search** button to open the Search configuration menu.



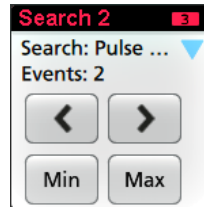
3. Use the configuration menu fields to set the search criteria in the same way that you would set for a trigger condition (select the **Search Type**, **Source**, and conditions on which to search).

**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**.*

---

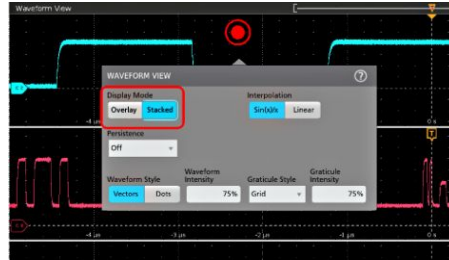
This opens the **Zoom** mode and moves the waveform to the previous or next event mark on the waveform.

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.

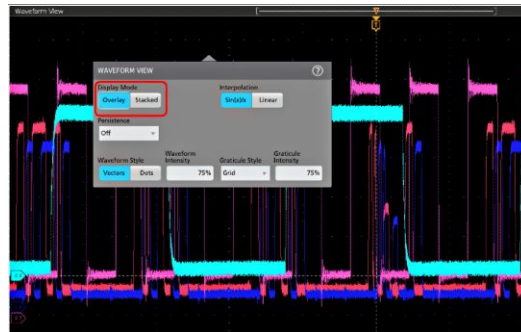
## Change waveform view settings

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.



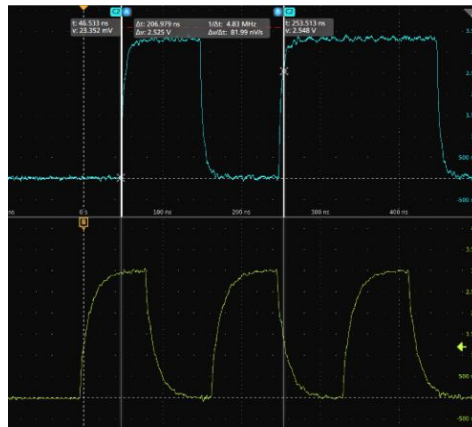
3. Use the other controls to set the waveform interpolation algorithm, waveform point persistence, style, and intensity, and graticule style and intensity.
4. Tap the **Help** icon on the menu title to open the Waveform View menu help topic for more information on the waveform view parameters.
5. Tap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

## Display and configure cursors

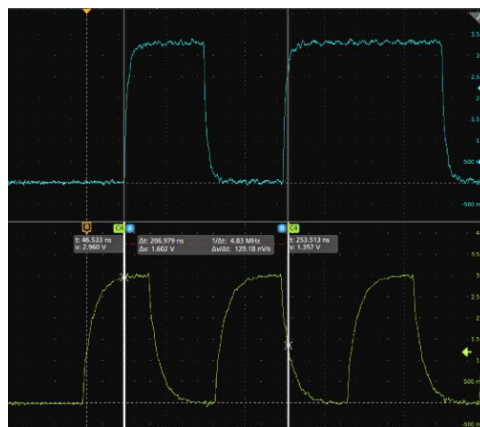
Cursors are on-screen lines that you can move to take measurements on specific parts of a waveform or plot, or between two different waveforms. Cursor readouts show both current position values and the difference (delta) between cursors.

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.

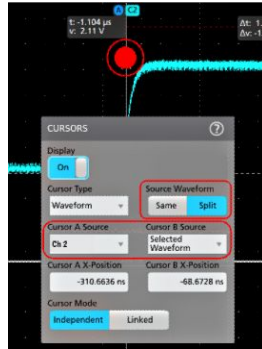
The cursors are added to the display.



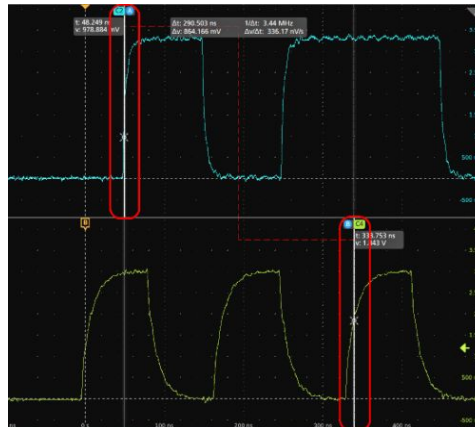
3. Use Multipurpose Knobs **A** and **B** to move the cursors, or touch and drag a cursor. Cursors show readouts that show position and difference measurements between the cursors.
4. To move the cursors to a different channel or waveform, just tap in that waveform graticule.



- 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.



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



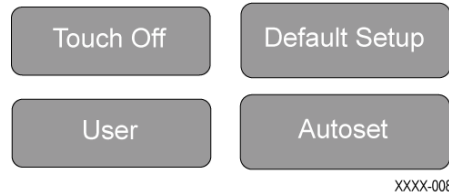
The cursors are moved to the specified waveforms.

- Tap the Help icon on the menu title for more information on the menu settings.
- 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).

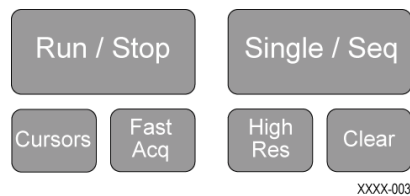


2. You can also select **File > Default Setup** to restore default settings.

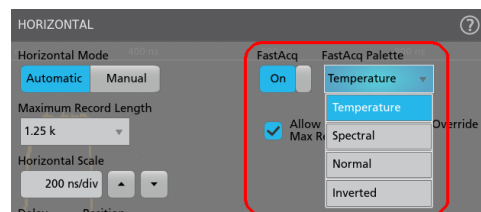
## Using Fast Acq

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 occurrences at intensity levels that reflect 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 with an intensity that reflects the 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.



---

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

---

---

**NOTE.** *Enabling FastAcq mode disables FastFrame mode (if it was enabled). Likewise, enabling FastFrame mode disables FastAcq mode (if it was enabled).*

---

## 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.

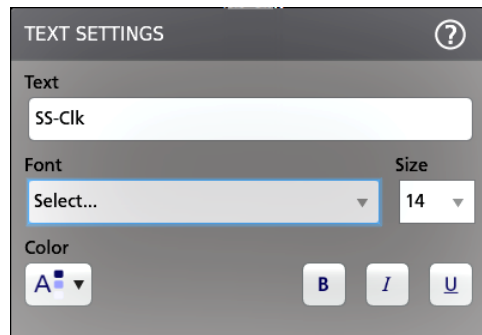
1. Tap the **Add New... Note** button. The oscilloscope adds a text placeholder object called Edit text to the Waveform view.

---

**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 to 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 137
- [Add a serial bus to the Waveform view](#) on page 139
- [Add a parallel bus to the Waveform view](#) on page 141

## 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. Use the upper **Label** field to enter a custom label for the overall digital channel. Double-tap on the field and use the virtual keyboard to enter label text. Or tap the field and use an attached keyboard to enter label text.
7. Tap a **Bit** control to toggle individual digital channel bits On or Off and remove them from the displayed logic waveform.
8. Tap a **Threshold** field and use multipurpose knob **A** to set individual channel bit threshold levels. You can also set the bit threshold by double-tapping the field and setting the threshold using the virtual keypad.
9. Use the bit **Label** fields to enter labels for the individual digital channel bits (D0-D7). Double-tap on the field and use the virtual keyboard to enter label text. Or tap the field and use an attached keyboard to enter label text.
10. Tap **Turn All Off** to turn all digital bits Off. Doing this closes the menu and removes the associated channel badge from the Settings bar.
11. To set all bit thresholds to the same value, enter a threshold value in the lower threshold field and tap **Set All Thresholds**.

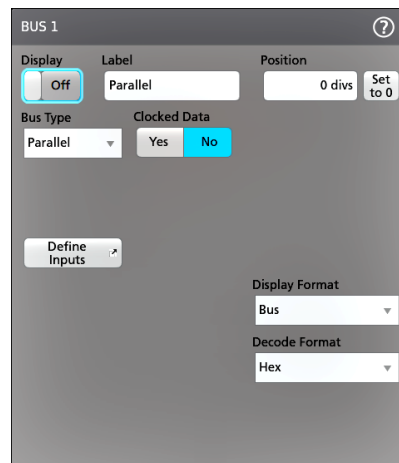
## Add a serial bus to the Waveform view

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

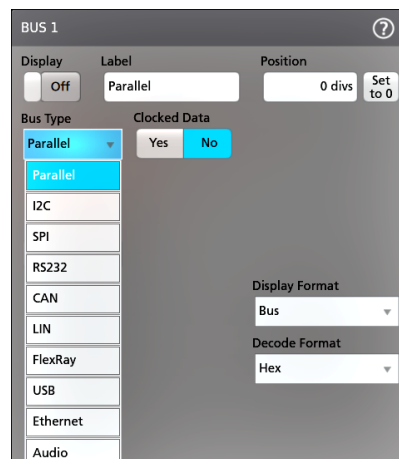
This instrument supports decoding parallel buses (standard with the instrument) and several serial buses options (see [Serial bus and trigger options](#) on page 19). All serial bus functions are options that must be purchased and installed before they are available in the instrument menus.

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

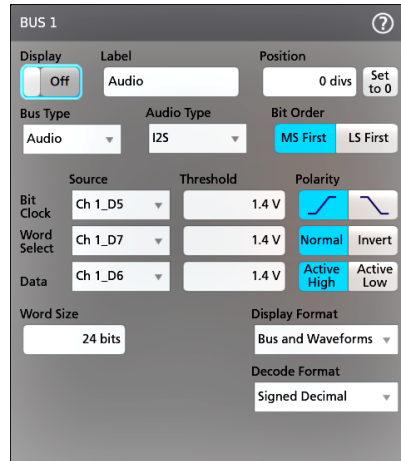
1. Tap the **Add New Bus** button on the Settings bar to add a Bus badge to the Settings bar, add a bus waveform to the screen, and open the Bus configuration menu. The default bus type is parallel.



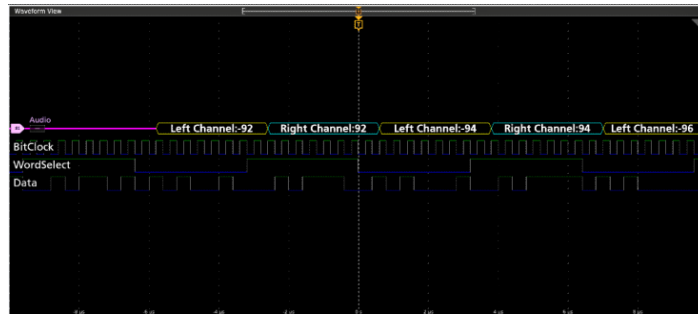
2. Tap **Bus Type** and select the bus type from the drop-down list.



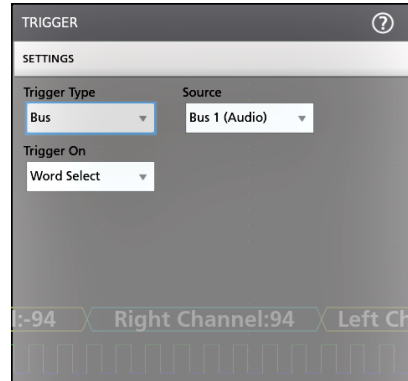
3. Use the fields and controls to select the bus signal sources, thresholds, other parameters, and the output format. The following example shows the settings for an Audio I<sup>2</sup>C serial bus.



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.



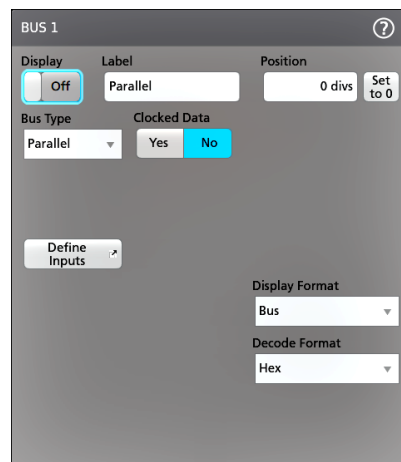
- For more information on serial bus settings, tap the Help button on the Bus configuration menu.

## Add a parallel bus to the Waveform view

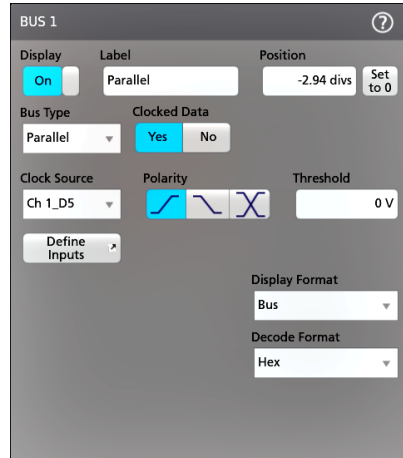
Use this procedure to add a parallel bus to the Waveform view.

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.

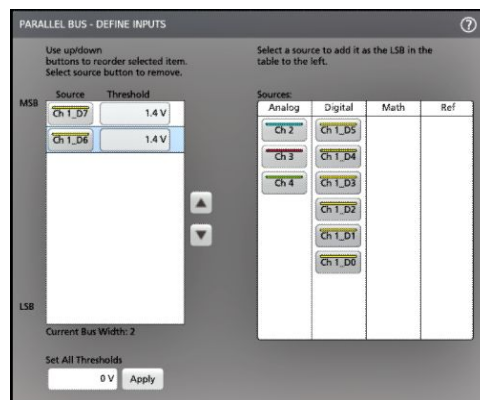
- Tap the **Add New Bus** button on the Settings bar. This adds a Bus badge bus to the Settings bar, adds a bus waveform to the screen, and opens the Bus configuration menu. The default bus type is parallel.



- If setting up a clocked bus:

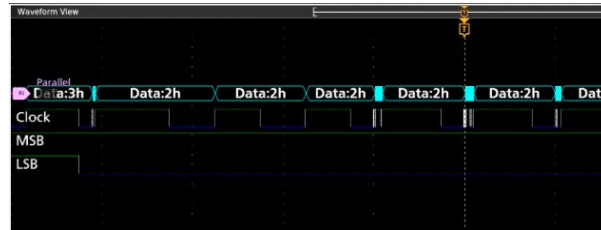


- a. Set Clocked Data to **Yes**.
  - b. Tap the **Clock Source** field and select the source for the parallel bus clock signal.
  - c. Tap the **Polarity** and **Threshold** controls and set the clock signal transition to detect and threshold level, respectively.
3. Tap **Define Inputs** and select 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

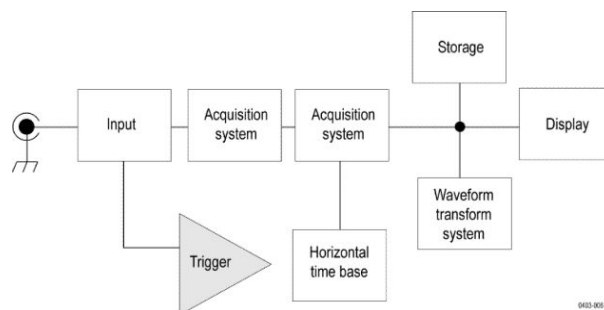
You can check the advanced trigger status in the trigger menu. The menu indicates the trigger type and then shows sources, levels, or any other parameters that are important for the particular trigger type. Use the following links for more information on advanced triggering.

- [Triggering concepts](#) on page 145
- [Set Trigger Holdoff](#) on page 153
- [Trigger on sequential events \(A and B triggers\)](#) on page 152
- [Set up trigger on a parallel bus](#) on page 151
- [Set up trigger on a serial bus](#) on page 151

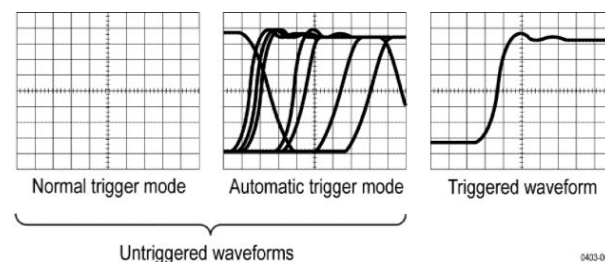
## Triggering concepts

**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).

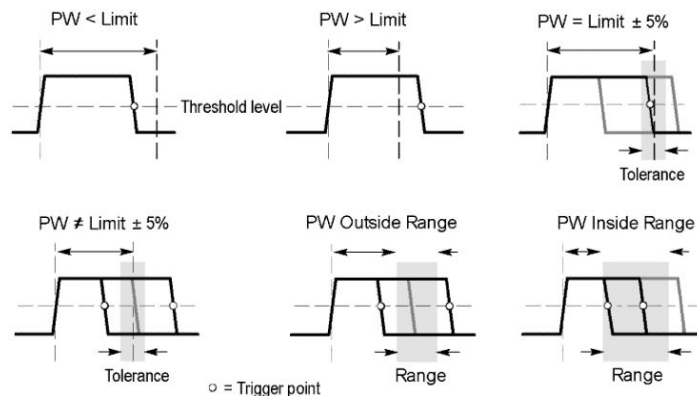
When a trigger event occurs, the instrument starts acquiring samples to build the posttrigger portion of the waveform record (displayed after, or to the right of, the trigger event). Once a trigger is recognized, the instrument will not accept another trigger until the acquisition is complete and the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

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Pulse Width**. See [Pulse Width Trigger configuration menu](#) on page 530.
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. See [Logic Qualification - Define Inputs configuration menu](#) on page 548.

## Trigger on a timeout event

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

To trigger on a timeout event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Timeout**. See [Timeout Trigger configuration menu](#) on page 545.
3. Tap **Source** and select the signal source.
4. Tap **Trigger When** and select the condition on which to trigger (**Stays High**, **Stays Low**, or **Either**).
5. Tap the **Threshold** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the threshold value that determines when the signal is high or low .
6. Tap the **Time Limit** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the maximum time that a signal can stay high or low before transitioning to the opposite state.
7. 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. See [Logic Qualification - Define Inputs configuration menu](#) on page 548.

## Trigger on a runt signal event

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

To trigger on a runt event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Runt**. See [Runt Trigger configuration menu](#) on page 535.
3. Tap **Source** and select the trigger source.
4. Tap **Trigger When** and select the condition on which to trigger.
5. Tap the **Upper Threshold** field and use the assigned multipurpose knob to set the threshold value that defines the upper limit that the signal must cross to be a valid signal transition (or double-tap the field and use the keypad to enter a value).
6. Tap the **Lower Threshold** field and use the assigned multipurpose knob to set the threshold value that defines the lower limit that the signal must cross to be a valid signal transition (or double-tap the field and use the keypad to enter a value).
7. If **Trigger When** is set to a limit value, use the **Time Limit** field to set the runt signal time constraints.
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. See [Logic Qualification - Define Inputs configuration menu](#) on page 548.

## Trigger on a signal window event

Use Window triggering to trigger when a signal enters or exits a specified threshold range (window), or remains inside or outside the window for a specified time limit.

To trigger on a window event:

1. Double-tap the **Trigger** badge to open the **Trigger** configuration menu.
2. Tap **Trigger Type** and select **Window**. See [Window Trigger configuration menu](#) on page 549.
3. Tap **Source** and select the signal source.
4. Tap **Trigger When** and select the condition on which to trigger.
5. Tap the **Time Limit** field to set the time value for the **Trigger When** condition.
6. Tap the **Upper Threshold** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the value that defines the upper window threshold.

7. Tap the **Lower Threshold** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the value that defines the lower window threshold.
8. If **Trigger When** is set to a limit value, use the **Time Limit** field to set the time constraints for how long a signal stays within or outside the window.
9. Use the **Threshold Crossing** field drop-down list to set the window threshold to cross when **Trigger When** is set to **Inside > Limit** or **Outside > Limit**.
10. 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. See [Logic Qualification - Define Inputs configuration menu](#) on page 548.

## Trigger on a logic event

Use the Logic trigger to trigger the oscilloscope when the specified logic conditions occur on any combination of analog and digital inputs.

To trigger on a Logic event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Logic**. See [Logic Trigger configuration menu](#) on page 527.
3. Set **Use Clock Edge?** to enable or disable finding logic patterns that occur on the clock edge of a specified channel.
4. Tap **Logic Pattern Define Inputs** and use the Logic Trigger menu to set the logic state and the signal logic threshold level for each analog or digital signal.
5. Tap **Clock Source** and select the signal to use as the clock.
6. Use the **Clock Edge** controls to set the signal transition edge to use to evaluate the logic condition at the clock transition.
7. Tap the **Clock Threshold** field and set 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).
8. Tap **Trigger When** and select the waveform condition on which to trigger when not using a clock edge.
9. Tap **Define Logic** and set the logic condition that must occur with all specified signals to cause a trigger event.

## Trigger on a setup-hold event

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 trigger on a Setup/Hold event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Setup & Hold**. See [Setup and Hold Trigger configuration menu](#) on page 542.
3. Tap **Clock Source** and select the signal to use as the clock.
4. Tap the **Clock Level** field and set 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).
5. Use **Clock Edge** to set which clock signal edge to use to start the trigger evaluation.
6. Tap **Define Inputs** and use the **Define Inputs** menu to set the signal input sources and their threshold levels. See [Setup and Hold Trigger - Define Inputs configuration menu](#) on page 545.
7. Tap the **Setup Time** field and set the length of setup time that the data signal must be stable before a clock edge occurs.
8. Tap the **Hold Time** field and set the length of hold time that the data signal must be stable after a clock edge occurs.

## Trigger on a rise or fall time event

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

To trigger on a Rise/Fall event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Rise/Fall**. See [Rise/Fall Time Trigger configuration menu](#) on page 532.
3. Tap **Source** and select the trigger signal source.
4. Tap **Trigger When** and select the condition on which to trigger.
5. Tap the **Time Limit** field to set the time value for the **Trigger When** condition.
6. Tap the **Upper Threshold** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the length of setup time that the data signal must be stable before a clock edge occurs.
7. Tap the **Lower Threshold** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the length of hold time that the data signal must be stable after a clock edge occurs.



8. Use the **Slope** controls to set the signal transition direction to detect. (rising, falling, or either direction).
9. To further refine the trigger conditions, set **Logic Qualification** to **On** and tap the **Define Inputs** button to define the logic conditions of other signals that must be true at the same time as the setup and hold event. See [Logic Qualification - Define Inputs configuration menu](#) on page 548.

## 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 19.

To trigger on a Rise/Fall event:

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. See [Bus Trigger configuration](#) on page 488.

## 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

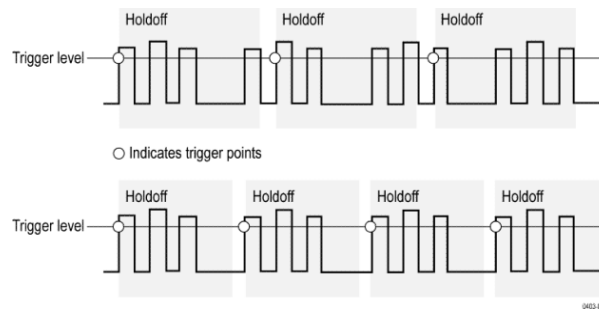
1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Sequence**. [Sequence Trigger configuration menu](#) on page 538
3. Set up the A Trigger Event:
  - a. Tap the **A Trigger Event** button to open the A Trigger Event menu.
  - b. Tap **Trigger 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 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 time 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 an external signal using the AUX input (MSO58LP, MSO64 only)

Use this procedure to trigger the instrument from an external signal connected to the AUX input (MSO58LP, MSO64 only).

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 oscilloscope Programmer Manual (Tektronix part number 077-1305-xx) for the correct command or commands to use.

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.** The *Aux* setting is only available for the *Edge* trigger type.

---

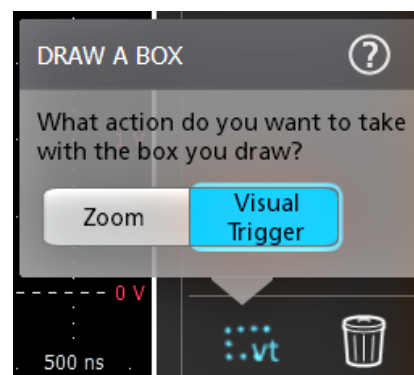
4. Set the values for **Coupling**, **Level**, and **Slope** to trigger on the Aux connector signal.

Settings take effect immediately.

## Create Visual Trigger areas

The Visual Trigger function extends standard triggering by letting you easily draw and define areas on the screen to further qualify trigger conditions. This procedure describes how to enable the visual trigger mode and draw areas on the screen.

1. Set the standard trigger conditions to trigger on and display the waveforms to which you want to add visual trigger areas.
2. Double-tap the **Draw-a-Box** icon (lower right of screen near the Garbage Can icon).
3. Tap the **Visual Trigger** button. This sets the Draw-a-Box function to Visual Trigger mode.



4. Tap outside the menu to close the **Draw A Box** menu. The Draw-a Box icon is highlighted to show that it is in Visual Trigger draw mode.
5. Draw a rectangle on the waveform screen for the first area to define. The area created is associated with the channel to which it was added (in Stacked mode) or the active channel (in Overlay mode), and assigned the default area logic condition of **In** (the area is true when the waveform intersects any part of the area).
6. Continue drawing rectangles to add other visual trigger areas:
  - a. If in Stacked mode, you can draw visual trigger areas in different waveform slices.
  - b. If in Overlay mode, select a channel before creating areas for that channel.
7. When you are done drawing visual trigger areas, single-tap anywhere to end the area draw function.
8. To draw areas again, tap the Draw-A-Box **vt** icon again.
9. To change the shape of an area, see [Edit visual trigger areas on the screen](#) on page 155 and [Edit visual trigger areas using the Area menu](#) on page 158.

## Edit visual trigger areas on the screen

Use the touchscreen or a mouse to directly edit an area, including moving, changing size, duplicating an area, moving individual vertices, adding and deleting vertices, and rotating the area.

1. To move a visual trigger area, touch and drag the area to a new position.

---

**NOTE.** You cannot drag a visual trigger area from one waveform slice to another while in Stacked display mode.

---

2. To delete a visual trigger area:
  - a. Touch and Hold or right-click on the area to delete.
  - b. Select **Delete Area** from the menu.
3. To resize a visual trigger area:
  - a. Single-tap an area to enable resize area mode. Resize mode draws resize points on all four sides and the corners of a boundary area that contains the area.



- b. Touch and drag a top, bottom, left, or right resize point to resize the area vertically or horizontally.
- c. Touch and drag a corner resize point to resize both the length and height of the area in the dragged direction.

---

**NOTE.** *You cannot move an area while in resize mode.*

---

- d. To exit resize mode, tap outside the visual trigger area. The instrument returns to normal operation.
4. To move an individual vertex in a visual trigger area:
- a. Tap the area twice to enable move vertex mode. The move vertex mode draws crosshairs on all vertices of the area.



- b. Touch and drag a vertex to move it to a new position.
  - c. To exit move vertex mode, tap outside the visual trigger area. The instrument returns to normal operation.
5. To add vertices to or delete vertices from a visual trigger area:
- a. Tap an area three times to enable add/delete vertex mode. Add/delete mode draws a crosshair at each existing vertex, and a plus symbol midway between each existing vertex.



---

**NOTE.** *A triangular area does not draw the existing vertices with crosshairs, as they cannot be deleted; only the midway plus symbols are shown to let you add vertices to a triangle.*

---

- b. Tap a vertex crosshair to delete that vertex.

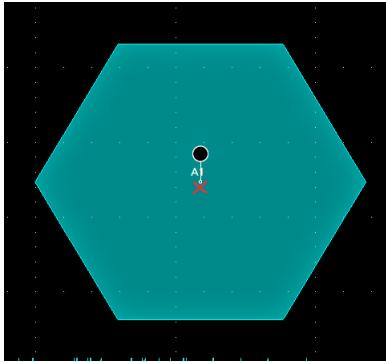
- c. Tap a plus symbol to add a new vertex at that location. The area is redrawn to show the new vertex. You can continue to add vertices; there is no limit to the number of vertices in an area.

---

**NOTE.** *You cannot move vertices while in add/delete vertex mode.*

---

- d. To exit add/remove vertex mode, tap outside the visual trigger area. The instrument returns to normal operation.
6. To rotate a visual trigger area:
- a. Tap four times in the area to enable rotate area mode.

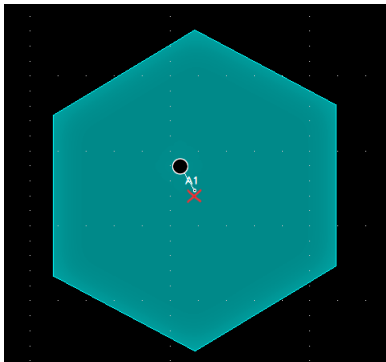


- b. Touch and drag the dot in the area to rotate the area.

---

**NOTE.** *Once you are touching and dragging the rotate dot, you can move your finger outside of the area to get finer control over the rotation increments.*

---



- c. To exit rotate area mode, tap outside the visual trigger area. The instrument returns to normal operation.

## Edit visual trigger areas using the Area menu

Use these procedures to edit Visual Trigger areas using the Area menu, including moving, changing size, moving individual vertices, adding and deleting vertices, and rotating the area.

See [Visual Trigger Area configuration menu](#) on page 555 for more information on the Area menu. See [Edit visual trigger areas on the screen](#) on page 155 for information on graphically editing an area on the screen.

1. To change the input source of an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Source** field.
  - c. Select a new source channel. The area moves to the selected source. If the selected source is not displayed, the instrument does not display that channel when the area is changed to that source.
  - d. Tap outside the menu to exit.
2. To change the shape of an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Shape** field.
  - c. Select the shape to which to change the area. The area changes shape immediately.
  - d. Tap outside the menu to exit.
3. To change the area waveform logic condition using the Area menu:

The waveform logic condition defines the logic true condition to assign an area when any part of a waveform intersects any part of an area:

**In** means that the area is true if any part of a waveform intersects any part of the area.

**Out** means that the area is true if a waveform never intersects any part of the area.

**Don't Care** ignores any waveform intersections. This creates a placeholder area to let you experiment with area definitions while developing your visual trigger.

  - a. Double-tap on the area to edit.
  - b. Tap the **Waveform Must Be** button of the logic true condition to assign to the area (**In**, **Out**, **Don't Care**). The selected logic condition is assigned immediately to the area.
  - c. Tap outside the menu to exit.



4. To change the height of an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Height** field and enter the height value as units of the current vertical channel settings (volts, amps, and so on). Or double-tap on the field and use the A knob to change the value. The shape immediately changes height while maintaining the vertical center position of the area.
  - c. Tap outside the menu to exit.
5. To change the vertical center of an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Vertical Center** field and enter the height value as units of the current vertical channel settings (volts, amps, and so on). Or double-tap on the field and use the A knob to change the value. The shape immediately changes the vertical center position of the area.
  - c. Tap outside the menu to exit.
6. To change the width of an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Width** field and enter the width value as horizontal time (seconds). Or double-tap on the field and use the A knob to change the value. The shape immediately changes width while maintaining the horizontal center position of the area.
  - c. Tap outside the menu to exit.
7. To change the horizontal center of an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Horizontal Center** field and enter the new horizontal center position value. Or double-tap on the field and use the A knob to change the value. The shape immediately changes the horizontal center position.
  - c. Tap outside the menu to exit.
8. To flip an area vertically using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Flip Vertical** button. The area immediately flips vertically around its center position.
  - c. Tap outside the menu to exit.
9. To flip an area horizontally using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Flip Horizontal** button. The area immediately flips horizontally around its center position.
  - c. Tap outside the menu to exit.

10. To rotate an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Rotate** field and use the **A** knob to change the value. The area immediately rotates when you enter a value or change the value with the **A** knob. Or double-tap on the field and enter the number of degrees to rotate the area, where zero (0) degrees is the position when the area was created. You can rotate an area from 0° to 360°.
  - c. Tap outside the menu to exit.
11. To add a point (vertex) to an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Edit Vertices** panel.
  - c. Select the vertex in the **Define Area** list counter-clockwise from you want to insert a new vertex. Selecting a vertex highlights that vertex on the area.
  - d. Tap **Insert Point**. A new vertex is added to the area, marked with an X, and is highlighted in the list.
  - e. Continue adding vertices as needed.
  - f. Tap outside the menu to exit.
12. To delete a point (vertex) from an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Edit Vertices** panel.
  - c. Select the vertex in the **Define Area** list that you want to delete. Selecting a vertex highlights that vertex on the area.
  - d. Tap **Delete Point**. A new vertex is added to the area, marked with an X, and is highlighted in the list.
  - e. Continue deleting vertices as needed.
  - f. Tap outside the menu to exit.
13. To move individual vertices in an area using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Edit Vertices** panel.
  - c. Select the vertex in the **Define Area** list that you want to move. Selecting a vertex highlights that vertex on the area.
  - d. Tap the X (time) axis or Y (Ampl.) field in the list and use the **A** and **B** knobs to change the position values. Or double-tap on each field and enter the new positional values for that vertex.
  - e. Tap outside the menu to exit.

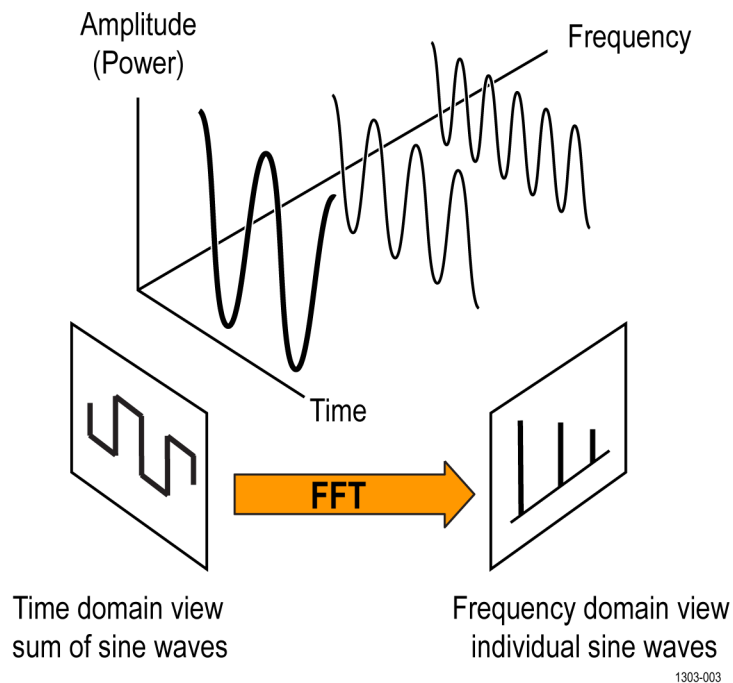
14. To reset the shape using the Area menu:
  - a. Double-tap on the area to edit.
  - b. Tap the **Edit Vertices** panel.
  - c. Tap **Reset Points**. The shape changes to a triangle centered in the display or slice.
  - d. Tap outside the menu to exit.



# Using Spectrum View

## Spectrum View concepts

Typical oscilloscope displays show electrical signals as a graph of time on the x-axis versus amplitude on the y-axis (time domain). A spectrum trace shows the same electrical signals as a graph of frequency on the x-axis versus amplitude (or magnitude) on the y-axis.



**Figure 11: The time and frequency domain components of a waveform.**

All time-domain signals can be defined as a composite of discrete sine waves, each with their own frequency, magnitude, and phase. The frequency-domain waveform (or spectrum trace) is a Fast Fourier Transform (FFT) decomposition of the time-domain signal into its constituent sine wave frequency components.

There are several advantages to examining signals in the frequency domain:

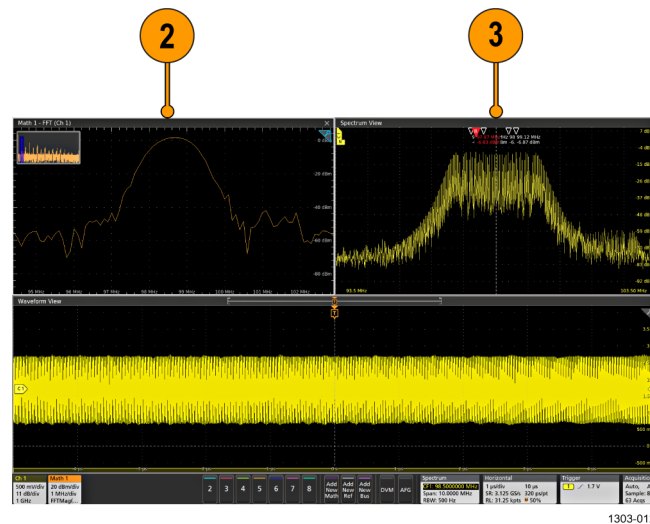
- It is easier to analyze signal noise characteristics. In the time domain, noise can appear as a fuzzy waveform, but in the frequency domain you can determine if it is broadband, random noise, or perhaps cross-talk from another signal on your DUT.
- It is easier to analyze and measure the distortions on signals. Although a 1% distortion on a sine wave may not be discernable in the time domain, it will be very obvious in the frequency domain as harmonics of the signal.
- Wireless communication measurements are described in the frequency domain in terms of frequency bands and channels. A spectrum trace lets you focus analysis on the relevant frequency range and make specialized measurements such as occupied bandwidth and modulation quality.

**FFT and Spectrum View**

Most modern oscilloscopes provide a Math FFT function to display the frequency-domain spectrum of a signal. The typical approach to creating an FFT is to use the same data acquisition used for the time-domain waveform (1), process it through an FFT, and display that data in a separate area or window (2).



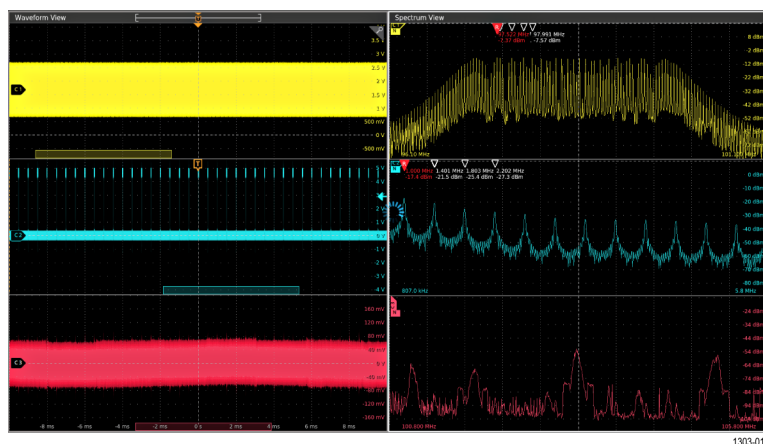
The disadvantage of a standard **Math FFT** is that the instrument uses the same acquisition data for both the time- and frequency-domain views. Therefore, the instrument settings for sample rate and record length affect both the time-domain and **Math FFT** acquisitions. Displaying an optimized FFT trace often involves manual calculations of the time-domain parameters like sample rate and record length. This also means that the instrument settings often cannot be optimized to show relevant information in both the waveform and **Math FFT** views at the same time. The following image shows a change in the Horizontal Scale setting that quickly changes the **Math FFT** plot.



1303-012

The **Spectrum View** feature in 5/6 Series instruments solves this problem by using a separate hardware path in the acquisition engine of each channel to acquire the frequency-domain data in parallel with the time-domain data. This frequency-domain data is processed, transformed, and displayed in a separate **Spectrum View** window (3) that is optimized to display and adjust spectrum traces independently of the time-domain waveform. Notice that the **Spectrum View** trace has not changed in the above images with changes to the time-domain signal, but the **Math FFT** signal is not usable for measurements in the second image.

Another key feature of **Spectrum View** is that you can display a spectrum trace for each **FlexChannel** input, along with their corresponding time-domain waveforms. The **Spectrum View** user interface is optimized to display and control frequency-domain parameters such as reference level, center frequency, span, and resolution bandwidth. Changing settings when the **Spectrum View** window is active (selected) does not change settings of the associated time-domain waveform in the **Waveform View**.



1303-016

## Display a spectrum trace

Spectrum traces are initially set up from each channel's badge menu.

1. Display the channel signal for which you want a spectrum trace.
2. Double-tap the channel badge.
3. Tap the **Spectrum View** panel.
4. Set **Display** to **On**. The instrument opens the **Spectrum View** window and adds a **Spectrum** badge to the Settings bar.
5. Select the **Frequency Domain Traces** to show in the **Spectrum View** (**Normal**, **Max Hold**, **Min Hold**, and/or **Average**).
6. Select the vertical **Units** (default is dBm).
7. Double-tap the **Spectrum** badge. See [Spectrum badge menu](#) on page 482.
8. Use the **Center Frequency**, **Span**, **FFT Window**, and **RBW Mode** controls to set these parameters to display the spectrum trace for the channel waveform. This menu also controls the **Spectrum Time Bar** (see [Spectrum View and Spectrum Time](#) on page 169).
9. To change the **Span**, use a horizontal pinch or expand motion on the touchscreen, or use the front-panel **Horizontal Scale** knob.

---

**NOTE.** Setting **Span** sets the same span value across all spectrum traces, regardless of the center frequency. For example, the trace 1 center frequency is set to 40 MHz, and span is set to 20 MHz, so trace 1 shows frequencies from 30 MHz to 50 MHz. If you add a spectrum trace for channel 2 with a center frequency of 60 MHz, trace 2 uses the same 20 MHz to show frequencies from 50 MHz to 70 MHz.

---

10. To change the vertical **Scale**, use a vertical pinch or expand motion on the touchscreen, or use the front-panel **Vertical Scale** knob.
11. To add peak markers to the trace, double-tap in a spectrum slice to open the **Spectrum View** menu and tap the **Markers** panel. Use the controls to set the number of markers to display, the threshold level at which to detect peaks, and the excursion range required by the peaks. See [Spectrum View window configuration menu](#) on page 478



## The Spectrum View user interface

A spectrum trace shows the magnitudes of the frequency components that make up the time-domain waveform. Use spectrum traces in the **Spectrum View** to view noise, distortion, and crosstalk information of time-domain signals.

### The Spectrum View elements

Predominant frequencies (with greater magnitude) are shown as peaks in the view. The vertical graticule represents the signal magnitude, and the horizontal axis represents frequency, with the starting (lowest) frequency at the left side of the graticule.

Each spectrum trace can display up to four spectrum trace types for each source channel: **Normal**, **Max Hold**, **Min Hold**, and **Average**. You can turn each of these traces on and off independently. You can display all or some of them simultaneously.

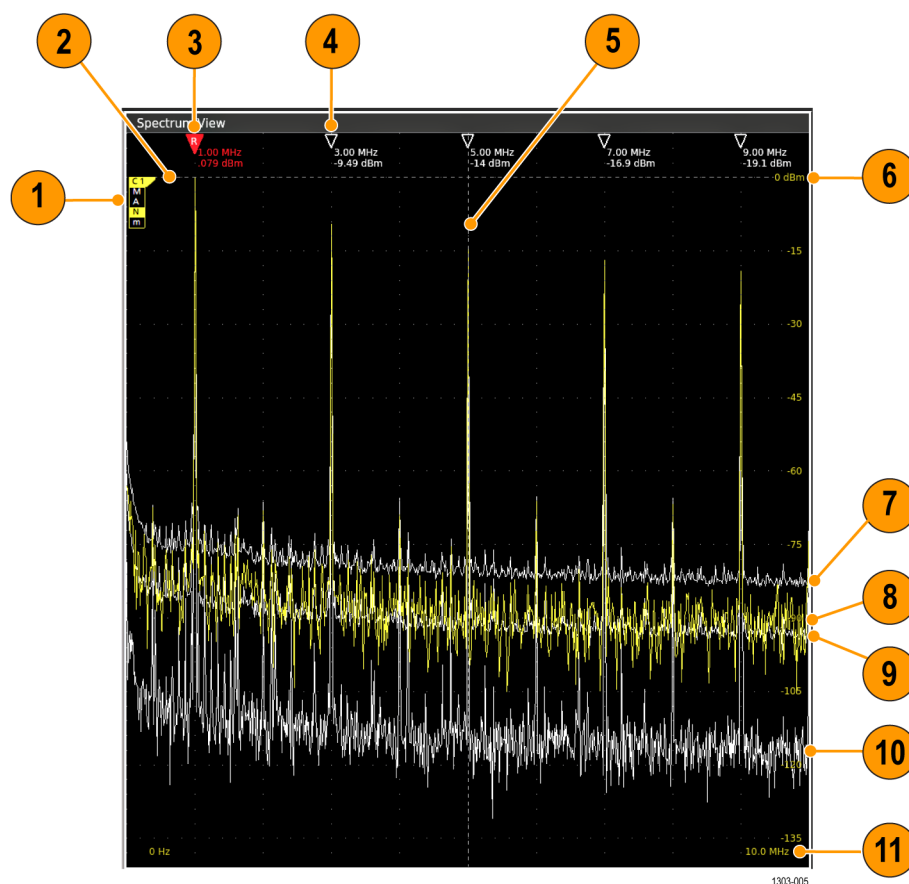


Figure 12: The Spectrum View window elements

1. The trace handle. See [Spectrum trace handle and trace types](#) on page 172.
2. The Reference level. The vertical scale (V/div) setting of the source time-domain waveform sets the spectrum trace reference level as follows:

Source vertical scale	Trace reference level	Source vertical scale	Trace reference level
500 $\mu$ V/div	-42 dBm	100 mV/div	4 dBm
1 mV/div	-36 dBm	200 mV/div	10 dBm
2 mV/div	-30 dBm	500 mV/div	18 dBm
5 mV/div	-22 dBm	1 V/div	24 dBm
10 mV/div	-16 dBm	2 V/div	30 dBm
20 mV/div	-10 dBm	5 V/div	38 dBm
50 mV/div	-2 dBm	10 V/div	44 dBm

3. The Reference marker labels and measures the highest-magnitude peak of the visible spectrum trace.
4. The Automatic markers label and measure the adjacent next-highest-magnitude peaks of the spectrum trace. The frequency and magnitude readouts can be absolute values or values relative to the Reference marker. See [Spectrum peak markers](#) on page 173.
5. The **Center Frequency** is a specified frequency at the center of the spectrum display. For many measurements it is the frequency of interest to measure, such as a carrier frequency. It may or may not be the highest magnitude peak in the trace. Each trace can have a separate center frequency setting. Changing the horizontal position of a trace in the **Spectrum View** window changes the center frequency of that trace and repositions the trace as needed.
6. The Amplitude (magnitude) scale. The vertical scale setting of the spectrum trace (pinch/expand actions on the touch screen, or the Vertical Scale knob on the front panel) sets the scale values.
7. The **Max Hold** trace shows the maximum data values that are accumulated and displayed over multiple acquisitions of the Normal trace.
8. The **Normal** trace shows each acquisition as new data is acquired. This is the default trace mode. The trace is shown in the same color as the source channel.
9. **Average** trace shows data that is averaged over multiple acquisitions. This is true power averaging, which occurs before the log conversion. Each power of 2 averaging reduces the displayed noise by 3 dB.
10. **Min Hold** trace shows the minimum data values accumulated over a specified number of acquisitions.
11. The **Span** is the range of frequencies (lowest to highest) to show on the spectrum trace. All spectrum traces have the same span regardless of the center frequency setting.

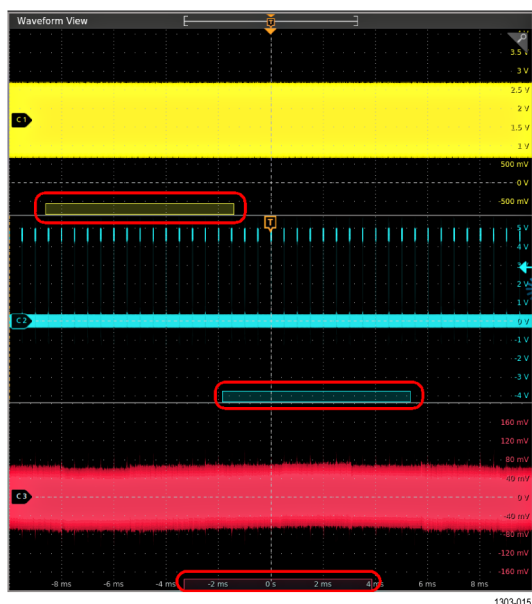
- Spectrum View guidelines**
- The **Spectrum View** window supports stacked and overlay display modes. When in Overlay display mode:
    - The vertical and horizontal in-graticule labels apply to the currently selected channel.
    - The MANm spectrum trace handle (see *Spectrum trace handle and trace types* on page 172) for the currently selected channel is shown highlighted, just like with analog channels.
    - The MANm trace handle for non-selected traces are shown in the same fashion as non-selected analog channels.
  - The **Spectrum View** does not support zoom.
  - Horizontal pinch and expand touch gestures adjust the spectrum span in the same style as a horizontal pinch/expand in the **Waveform View**.
  - A horizontal pan adjusts the center frequency. While panning, the pan motion will pause momentarily when the Reference Marker reaches the middle of the graticule. This provides a very simple Marker to Center functionality.
  - Vertical pinch and expand touch gestures adjust the spectrum vertical scale in the same style as a vertical pinch/expand in the **Waveform View**.
  - A vertical pan adjusts the vertical position. While panning, the pan motion will pause momentarily when the trace handle reference level reaches the top of the graticule. This provides a very simple reference level to top functionality.

## Spectrum View and Spectrum Time

Spectrum Time controls the area in the time-domain acquisition used to create the **Spectrum View** trace. Conceptually like gating, use Spectrum Time to examine the spectrum of specific areas of interest in the acquisition record.

**Spectrum Time** is the part of the time-domain waveform acquisition used to create the **Spectrum View** trace for that channel.

**Spectrum Time** is shown by a shaded bar along the bottom of a Channel graticule in the **Waveform View**. This bar is called the **Spectrum Time Bar**. The Spectrum Time Bar visually indicates both the range and time position in the acquisition record used to create the spectrum trace.



The Spectrum Time width (amount of the acquisition record) is determined by the FFT Window factor divided by Resolution Bandwidth (RBW). Changing the FFT window type and/or the RBW parameters changes the Spectrum time width.

The Spectrum Time position can be moved through the time domain acquisition using a variety of methods including tapping and dragging it, using the **Wave Inspector Pan** knob, or with the **Spectrum Time** position control in the **Spectrum** configuration menu.

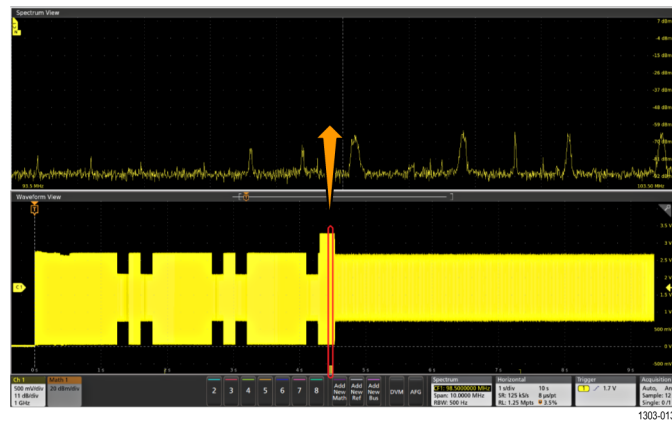
The FFT Window, **RBW**, and **Spectrum Time** settings are in the **Spectrum** menu (see [Spectrum badge menu](#) on page 482).

The Spectrum Time Bar is shown whenever a **Spectrum View** trace is enabled for a channel.

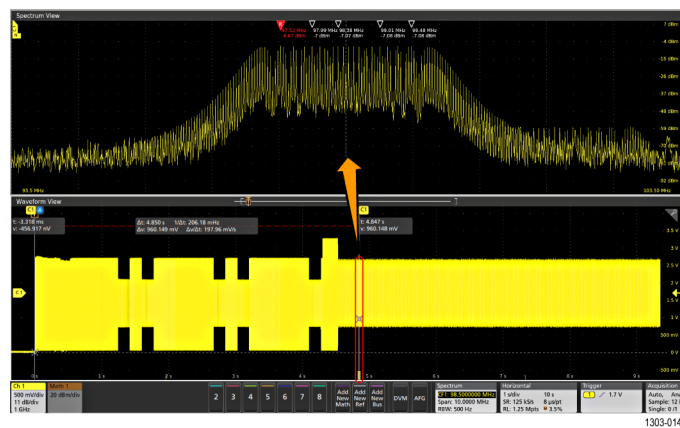
**Example use of Spectrum Time to determine the time to a stable signal.**

This example shows the use of Spectrum Time to determine the time for a spread spectrum (SS) clock to become stable after power-up. The clock signal is centered around 98.5 MHz.

An acquisition was taken with a long enough time setting to ensure that the SS clock signal had stabilized. The RBW was set to 500 Hz, and the Spectrum Time position was set to just before the SS clock signal became stable. See the following image.



The Spectrum Time Bar position was slowly moved to the right until a valid SS clock signal was shown in the Spectrum view, as shown in the following image.

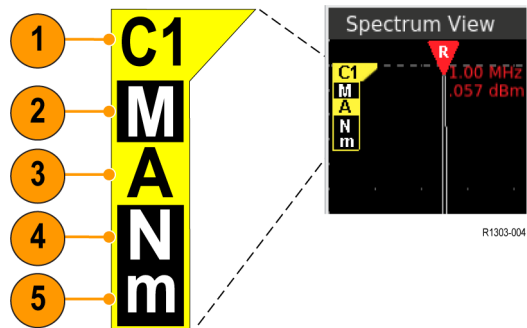


Using cursors in the **Waveform View**, the measured time from power-up to when the SS clock signal is stable was 4.847 seconds.

- Spectrum Time guidelines**
- Spectrum Time can be as long as, but never longer than, the RF acquisition time.
  - To position the Spectrum Time Bar to areas of interest, tap and drag the bar, or use the front-panel **Pan** knob to move the bar of the selected Waveform View channel.
  - You can set the Spectrum Time Bars to lock together across all input channels, so that moving one moves all Spectrum Time bars. The Spectrum Time Bars can also be unlocked and moved to show the Spectrums of different portions of different channels.
  - When **Spectrum Time** is not locked across all channels (**Lock Spectrum Time Across All Channels** is not selected), and zoom is **On**, the **Pan** front-panel knob controls the **Zoom Box** of the selected channel.
  - When **Spectrum Time** is not locked across all channels (**Lock Spectrum Time Across All Channels** is not selected), and zoom is **Off**, the **Pan** front-panel knob controls the **Spectrum Time Bar** of the selected channel.

## Spectrum trace handle and trace types

The spectrum trace handle provides details on the trace reference level, the source channel for the trace, and which trace types are displayed and selected.



**Figure 13: The Spectrum View trace handle elements**

1. The trace's source channel (at top of the handle) and trace reference level (top edge of the handle, with dashed line extending across the screen). Spectrum View shows the spectrum traces relative to a Reference Level reference point. The color of the handle is the same as the source channel.  
If the Reference level is outside of the displayed trace (above or below the graticule), the handle is drawn with the channel source text pointing toward the Reference level.
2. A capital **M** indicates that the maximum trace is enabled. If the maximum trace is not enabled, this letter is not shown.
3. A capital **A** indicates that the average trace is enabled. If the average trace is not enabled, this letter is not shown.
4. A capital **N** indicates that the normal trace is enabled. If the normal trace is not enabled, this letter is not shown.
5. A small **m** indicates that the minimum trace is enabled. If the maximum trace is not enabled, this letter is not shown.

The above image shows all traces enabled and the **A (Average)** trace selected. Highlighting around a letter indicates that that trace type is selected.

### Trace handle and trace type guidelines

There are important distinctions between enabled (displayed) and selected traces:

- An enabled trace is a trace type that is enabled in the **Spectrum View** panel of the source channel badge and is shown in the spectrum view window. If the trace type is not enabled, its corresponding letter is not shown in the trace handle and there is no corresponding trace shown on the screen.
- A selected trace (highlighted around the letter) is the trace that is used for marker and cursor readouts. Only one trace type can be selected at a time. Double-tap the trace handle to change the selected trace type.
- You can enable and display any combination of trace types for each spectrum trace.
- Disabling all trace types causes the spectrum view for that channel to close.
- If there is only one spectrum trace shown in the **Spectrum View**, and you disable all trace types or turn off the spectrum function for that channel, the **Spectrum View** window closes.

## Spectrum peak markers

Automatic peak markers assist with quickly identifying the frequency and amplitude of peaks in the spectrum trace. Access the marker settings by double-tapping in a spectrum trace slice window (Stacked mode) or trace (Overlay mode) to open the **Spectrum View** menu.

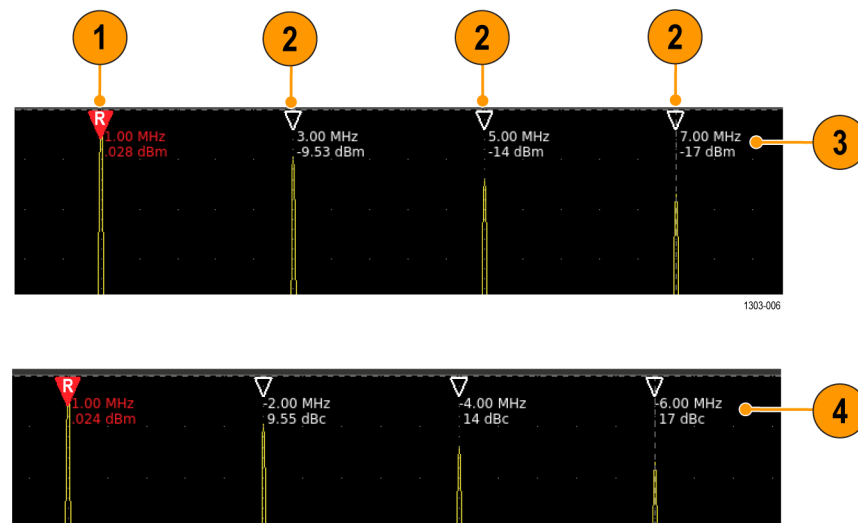


Figure 14: The spectrum peak markers

1. The highest peak shown is considered the **Reference** peak, and is labeled with a red Reference marker triangle. The **Reference** marker readout (also in

red) always shows the absolute amplitude and frequency of that peak, regardless of the Readout type setting (**Absolute** or **Delta**).

2. The **Automatic** peak markers are hollow white triangles that mark the next highest peaks in the trace, depending on the marker parameters.
3. **Absolute** readouts on markers show the actual frequency and amplitude values of the marked peaks of the selected trace.
4. **Delta** readouts on Automatic markers show the delta frequency and delta amplitude of the marked peaks, of the selected trace, relative to the Reference marker readout.

### Marker parameters

To set marker parameters, double-tap in the **Spectrum View** window and tap the **Markers** panel (see [Spectrum View window configuration menu](#) on page 478). Marker parameters include:

- Turning marker display **On** or **Off**.
- The number of markers to display (1 to 11). Selecting 1 shows just the **Reference** marker.
- The peak **Threshold** value, which is the minimum absolute peak level that peaks must equal or exceed to display a mark on that peak.
- The peak **Excursion** value, which is the minimum magnitude (excursion range) that peaks must equal or exceed to display a mark on a peak.
- The type of readout values to display (**Absolute** or **Delta**).

## Spectrum View and front panel controls

The front panel controls provide direct access to many spectrum-related parameters when you select the **Spectrum View** window.

- The **Vertical Scale** knob controls the vertical scale of the currently selected spectrum trace. Note that this is purely a display function. It does not change the source channel's volts/div setting or the reference level of the spectrum.
- The **Vertical Position** knob controls the vertical position of the currently selected spectrum. This is a display function to visually position the spectrum trace.
- Pushing the **Vertical Position** knob resets the vertical position such that the spectrum reference level is at the top of the slice (**Stacked** mode) or the top of the display (**Overlay** mode).
- The front panel vertical channel buttons only function for time domain (analog) channels. If you are in **Spectrum View** mode, pushing a channel button activates the **Waveform View** window and selects the button's channel.
- The **Horizontal Scale** knob controls span. This affects all spectrum traces.

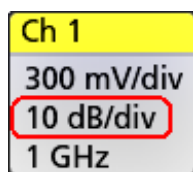


- The **Horizontal Position** knob controls center frequency. This affects all channels if center frequencies are locked. If not, it affects only the currently selected spectrum trace.
- Pressing the **Horizontal Position** knob moves the Reference Marker to center screen.

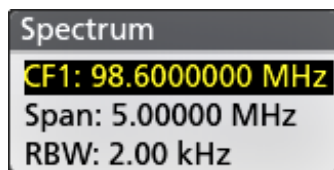
## Spectrum View and badges

Spectrum View makes changes to channel badges and adds a new Spectrum badge.

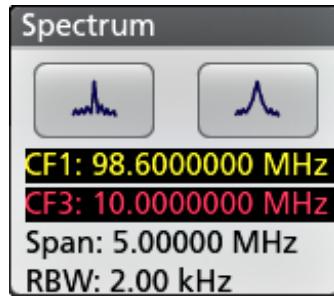
When a channel is set to show a **Spectrum View** trace, the second row of the channel badge changes from the probe symbol or input impedance readout to the vertical units of that channel's spectrum trace.



When the first spectrum trace is enabled for any analog channel, the instrument opens the **Spectrum View** window and adds the **Spectrum** badge to the **Settings Bar**. The first line of the Spectrum badge shows the center frequency of the selected spectrum trace.

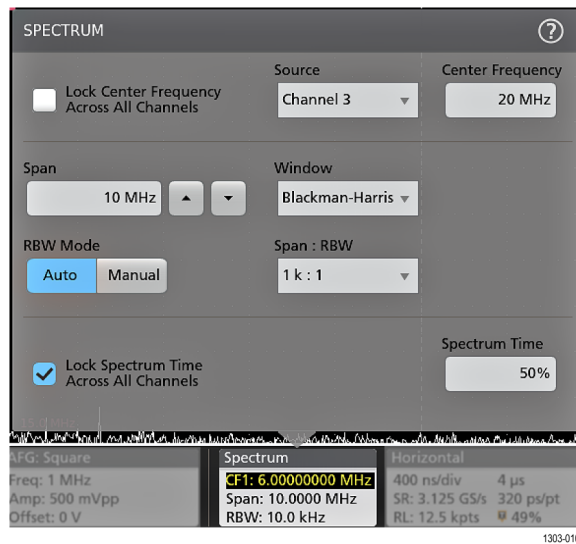


Tap the **Spectrum** badge to show the **Span** buttons to quickly increase or decrease the Spectrum trace span. Changing the span for one Spectrum trace changes the span for all traces.



When more than one Spectrum trace is enabled, the Spectrum badge lists the enabled Spectrum traces.

Double-tap the **Spectrum** badge to open the **Spectrum** menu to set Span, RBW, and other parameters.



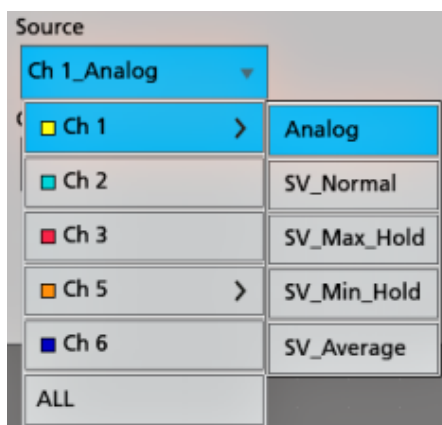
The spectrum **Span**, **RBW** (resolution bandwidth), and (FFT) **Window** settings apply to all spectrum traces. The **Center Frequency** and **Spectrum Time** can be set for each Spectrum trace. See the [Spectrum badge menu](#) on page 482.

## Save and recall spectrum traces

You can save spectrum trace data to a file, and recall the trace to use as a reference trace or for other analysis purposes. Recalled spectrum traces are shown in the **Spectrum View** window.

**Save a spectrum trace** Use the **File > Save As** menu to save spectrum trace data to a specified location. See [Save a waveform or spectrum trace to a file](#) on page 194.

In the **Source** field, select the spectrum trace source channel and select the spectrum trace type from the expanded list. Trace types (Normal, Max Hold, and so on) must be enabled in the spectrum trace to be listed in the **Source** menu.



**Recall a spectrum trace** Use the **File > Recall** menu to navigate to and select a spectrum trace file to recall. See [Recall a reference waveform or spectrum trace](#) on page 198. Select the spectrum trace file to recall in the **Name** field.

Spectrum trace file names include the name of the source channel (rf1 = Ch1, rf2 = Ch2, and so on) and the spectrum trace type (normal, average, and so on). For example, the file name SpreadSpectrum\_022\_rf5\_average.wfm indicates that the spectrum trace was from channel 5, and is an averaged spectrum trace.



---

# 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 \(Stacked or Overlay\)](#) on page 179
- [Set the Waveform Interpolation mode](#) on page 180
- [Set the waveform persistence, style, and intensity](#) on page 180
- [Set the graticule style and intensity](#) on page 181

## 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

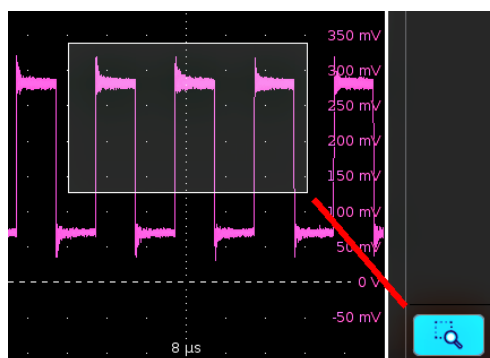
Use the zoom tools to magnify waveforms to view signal details.

## Turn on Zoom mode

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

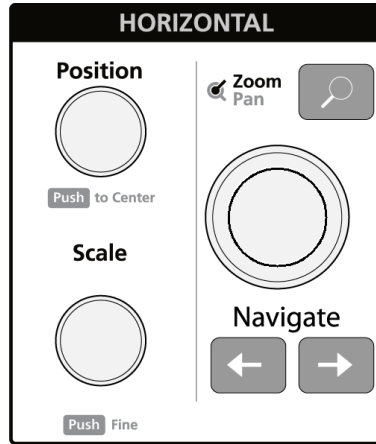
To enable Zoom mode, use one of the following methods:

1. Double-tap the **Draw a Box** button and select the **Zoom** button. Tap outside of the menu to close it.

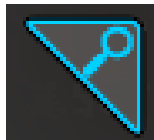


Tap the Draw-a-Box button, then touch and drag in the Waveform view to draw a box on the area of interest. The Draw a Box Zoom mode stays enabled so that you can draw boxes to zoom in different areas of the Zoom Overview, or draw boxes to further zoom in on the same area.

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



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



4. Zoom overview:
  - a. To use Zoom once enabled, tap the Draw a Box icon touch and draw a box around an area of interest in the Waveform or Plot view to immediately display the zoomed waveform and the Zoom Overview window.
  - b. You can draw boxes in the Waveform view, most plots, and the Zoom Overview area.
  - c. The **Draw a Box** Zoom mode stays enabled so that you can draw boxes to zoom in different areas of the Zoom Overview, or draw boxes to further zoom in on the same area.
  - d. Single-tapping anywhere disables the zoom box draw mode.
  - e. To exit the zoom display mode, tap the Zoom icon in the corner of Waveform and Plot views, or push the front-panel **Zoom** button.
5. For more information on zoom:
  - [The Zoom user interface elements](#) on page 74
  - [Zoom mode and Searches](#) on page 185
  - [Using Wave Inspector front-panel controls for zoom](#) on page 185

## 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 74.

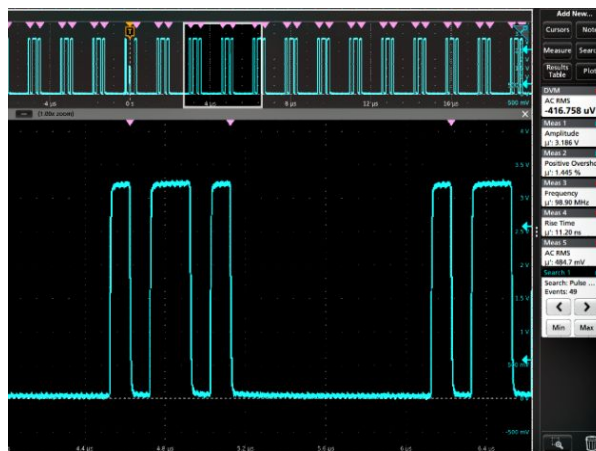
## Zoom mode and Searches

Use Zoom and Searches to find events of interest on a waveform.

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 in the Zoom Overview and Zoom View window to the previous or next search mark on the waveform. Push and hold a front-panel navigation button to continuously step to and show the next search mark.

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



For information on creating a Search, see [Add a Search](#) on page 127.



---

# Customizing measurements

After adding a measurement, you can customize the measurement for more precise results by using gating, setting reference levels, adding a filter, limiting the results to view, 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 228.

See the following topics for more information.

- **Label** your measurement to clarify documentation. See [Label a measurement](#) on page 187.
- **Reference levels** determine how time-related measurements are taken. See [Set measurement reference levels](#) on page 188.
- **Gating** confines the measurement to a certain portion of a waveform. See [Set measurement gates](#) on page 190.
- **Filters** control the band pass of a measurement. See [Set measurement filters](#) on page 191.
- **Limiting** the results you can specify a minimum or maximum value. See [Set measurement limits](#) on page 192.

## 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 228.

Prerequisite: To set measurement reference levels you must be taking a measurement. See [Add a measurement](#) on page 112.

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 228.

Prerequisite: To set measurement reference levels you must be taking a measurement. See [Add a measurement](#) on page 112.

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 228.

To set measurement gates you must be taking a measurement. See [Add a measurement](#) on page 112.

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 228.

To set measurement filters you must be taking a measurement. See [Add a measurement](#) on page 112.

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 228.

To set measurement limits you must be taking a measurement. See [Add Measurements configuration menu overview](#) on page 210.

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 or spectrum trace 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. You can also save Spectrum View traces to files.

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.

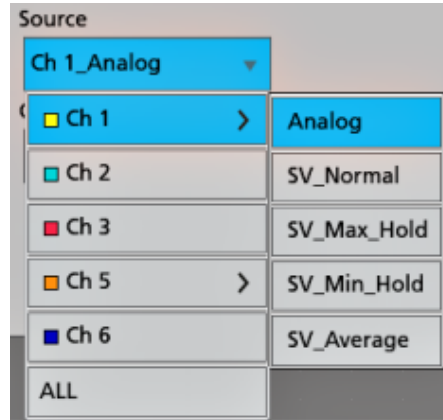
5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.

- 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.

The **Source** menu shows a right arrow in the source list if a source has more than one waveform available to save, such as an analog channel and its corresponding spectrum trace. Tap the source menu item to expand that item to show available waveform sources.



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, measurements, reference waveforms of active signals, plots, and eye diagram masks if enabled.

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 or spectrum trace

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 to the last location used to recall a waveform or spectrum trace file. If the location is not available, the menu defaults to the instrument C drive.

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

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

3. Tap the file name in the list to add the file to the **File Name** field.
4. Tap **Open** to exit the menu and add the reference waveform to the screen.



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

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

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

---

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

---

7. Tap **Recall**.

The reference waveform is loaded and displayed, and a **Ref** badge is added to the Settings bar. If you recalled a spectrum trace waveform, that waveform is shown in the **Spectrum View** window, with the handle name of R1, R2, and so on.

## 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.

## Open a mask test file for an Eye Diagram plot

Use this procedure to open (load) and display an eye diagram mask for an eye diagram plot.

Prerequisites:

An eye diagram plot. To display an eye diagram plot, see [Display an Eye Diagram plot](#) on page 120.

An eye diagram mask file that defines at least three segments.

An eye diagram mask file that matches the time scale (unit interval) and vertical settings of the eye diagram.

1. Double-tap anywhere in the eye diagram plot to open the plot configuration menu.
2. Tap the **Mask** panel.
3. Tap **Browse**.
4. Use the **Browse Mask File Location** menu to navigate to and select an eye diagram mask file. See [Recall Mask File configuration menu](#) on page 447.
  - If the file was recently recalled, tap the arrow in the **File Name:** field and select from the list of recently recalled files.

---

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

---

5. Tap **Recall** to close the menu and display the three eye diagram masks in the eye diagram plot.

---

# 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
<b>Run/Stop</b>	Toggles the oscilloscope between constant acquisition (Run) and no acquisitions (Stop). When stopped, the oscilloscope shows waveforms from the last completed acquisition.
<b>Single/Seq</b>	Acquires a single acquisition or a set number of acquisitions, then stops.
<b>Clear</b>	Erases acquired waveform data points from memory. Applies to all live acquisition waveforms.

Field or control	Description
<b>Acquisition Mode</b>	<p><b>Sample</b> 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.</p> <p><b>Peak Detect</b> 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.</p> <p><b>High Res</b> 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.</p> <p>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.</p> <p>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.</p> <p><b>Envelope</b> 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.</p> <p><b>Average</b> acquires and displays a waveform record that is the average result of several acquisitions. This mode reduces random noise.</p>
<b>Single Sequence/Stop After</b>	Enables stopping acquisitions after a specified number of acquisitions. Only works when using the Single/Sequence button.
<b>Save on Trigger</b>	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 <b>Configure</b> button to set save location.
<b>Configure</b>	Opens the <b>Save On Trigger</b> menu to set the type of file to save and the save location. See <a href="#">Save On Trigger menu</a> on page 205.

Field or control	Description
<b>Timebase Reference Source</b>	<p>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.</p> <p>On some instruments External is divided into External (<math>\pm 2</math> ppm) and External (<math>\pm 1</math> k ppm) selections. The narrow External (<math>\pm 2</math> ppm) setting locks the timebase within 2 ppm of 10 MHz. The wide External (<math>\pm 1</math> k ppm) allows the timebase to be within 1 k ppm of 10 MHz.</p>
<b>Fast Frame panel</b>	Sets the Fast Frame acquisition mode. See <a href="#">FastFrame Panel</a> on page 206

### 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
<b>Save Location</b>	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.
<b>Browse</b>	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.
<b>File Name</b>	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.
<b>Files to Save</b>	Select the file type(s) to save (Waveform, Screen Capture, or both).
<b>Format</b>	Select the file format to which to save the waveform or screen capture.
<b>Source</b>	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
<b>FastFrame</b>	Enables or disables the FastFrame mode. When enabled (On), the screen shows the last acquisition made when FastFrame was enabled.
<b>Number of Frames</b>	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.
<b>Summary Frame</b>	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.**

- Enabling FastFrame mode disables FastAcq mode (if it was enabled). Likewise, enabling FastAcq mode disables FastFrame mode (if it was enabled).
- You should 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.
<b>Selected Frame</b>	<p>Shows the selected frame in all displayed waveforms (analog, digital, and math). Use Multipurpose A knob to scroll through and select specific frames.</p> <p>To enter a specific frame number to display, use the Selected Frame field in the <a href="#">FastFrame configuration menu</a>.</p> <p>Frame scrolling uses a wrap-around method. For example, when you reach the end of the frame list, scrolling further displays frame 1.</p> <p>If you have enabled overlay mode, the current frame's waveform is highlighted in blue in all displayed waveforms (analog, digital, and math).</p>
<b>Reference Frame</b>	<p>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.</p> <p>To directly enter a specific frame number to display, use the <b>Reference Frame</b> field in the <a href="#">FastFrame configuration menu</a>.</p> <p>Only available when <b>Include Reference Frame in Badge</b> is selected in the FastFrame configuration menu.</p>
Time stamps	<p>Shows the time difference between frame one and the Selected frame. If <b>Include Reference Frame in Badge</b> 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.</p> <p>When both Selected and Reference frames are enabled, a <b>Delta</b> time readout shows the time difference between the Selected frame and the Reference frame.</p> <p>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.</p>
Navigation buttons	Tap to display the previous or next frame. Navigation buttons are only displayed when acquisition is set to Stopped.
<b>Summary</b>	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.
<b>Overlay Frames</b>	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).
<b>Selected Frame</b>	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 <b>Selected Frame</b> field and use the virtual keypad to enter a frame number.
<b>Include Reference Frame in Badge</b>	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
<b>Reference Frame</b>	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 <b>Selected Frame</b> field and use the virtual keypad to enter a frame number.  Only available when <b>Include Reference Frame in Badge</b> is selected.
<b>Plot FastFrame Results</b>	Opens and displays the selected plot type.  <b>Timestamp Time Trend</b> 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 228.

### Add Measurements menu fields and controls

Field or control	Description
<b>Measurement tabs</b>	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 <a href="#">The Jitter tab (Advanced Jitter and Eye Analysis) (option 5-DJA, 6-DJA)</a> on page 217. See <a href="#">The Power tab (optional)</a> on page 222.
<b>Measurement description</b> (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.
<b>Source</b>	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.
<b>Add button</b>	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
<b>Amplitude Measurements</b> panel	Tap the panel bar to list the available amplitude measurements. Touch and drag the list to scroll through all measurements. See <a href="#">Amplitude Measurements panel</a> on page 212.
<b>Timing Measurements</b> panel	Tap the panel bar to list the available time measurements. Touch and drag the list to scroll through all measurements. See <a href="#">Timing Measurements panel</a> on page 214.
<b>Jitter Measurements</b> panel	Tap the panel bar to list the available standard jitter measurements. See <a href="#">Jitter Measurements panel</a> on page 217.  <b>NOTE.</b> If you have installed Advanced Jitter and Eye Analysis (option 5-DJA, 6-DJA), the <b>Jitter Measurements</b> panel is removed from the <b>Standard</b> measurement tab and is replaced with a <b>Jitter</b> tab at the top of the <b>Add Measurements</b> menu. See <a href="#">The Jitter tab (Advanced Jitter and Eye Analysis) (option 5-DJA, 6-DJA)</a> on page 217.

### Other tab measurements (require optional licenses)

Add Measurement tab	Description
Jitter	Advanced Jitter and Eye Analysis measurements (option 5-DJA, 6-DJA). Provides triggers and measurements for advanced jitter and eye analysis. See <a href="#">The Jitter tab (Advanced Jitter and Eye Analysis) (option 5-DJA, 6-DJA)</a> on page 217.
Power	Advanced Power Analysis (option 5-PWR, 6-PWR). Provides measurements for input analysis, output analysis, amplitude analysis, timing analysis, and switching analysis. See <a href="#">The Power tab (optional)</a> on page 222.
DDR	DDR (option 6-DBDDR3). Provides measurements for DDR amplitude and timing analysis. See <a href="#">The DDR measurements tab (Requires option SUP6-DBDDR3)</a> on page 226.

### 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 212
  - [Timing Measurements panel](#) on page 214
  - [Jitter Measurements panel](#) on page 217
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, all jitter measurements are in the **Jitter** tab. See [The Jitter tab \(Advanced Jitter and Eye Analysis\)](#) (option 5-DJA, 6-DJA) on page 217.*

---

## 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
<b>AC RMS</b>	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.
<b>Amplitude</b>	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.
<b>Area</b>	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.
<b>Base<sup>1</sup></b>	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.
<b>Maximum</b>	The maximum data point value. You can take this measurement on each cycle in the waveform record or across the entire waveform record.
<b>Mean</b>	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.
<b>Minimum</b>	The minimum data point value. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
<b>Negative Overshoot<sup>1</sup></b>	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.
<b>Peak-To-Peak</b>	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.
<b>Positive Overshoot<sup>1</sup></b>	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.
<b>RMS</b>	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.
<b>Top<sup>1</sup></b>	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.

<sup>1</sup> Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated.

**See also.** [Measurement configuration menu overview](#) on page 228

### **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
<b>Burst Width<sup>2</sup></b>	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.
<b>Data Rate<sup>2</sup></b>	Data Rate is the reciprocal of Unit Interval. The measurement is taken on each bit of the waveform record.
<b>Delay</b>	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.
<b>Duration N-Periods<sup>2</sup></b>	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.
<b>Fall Time</b>	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.
<b>Falling Slew Rate</b>	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.
<b>Frequency<sup>2</sup></b>	The frequency of the waveform. Frequency is the reciprocal of Period (Frequency = 1/Period).
<b>High Time</b>	The time the signal remains above the Top reference level. The measurement is taken on each cycle in the waveform record.
<b>Hold Time<sup>2</sup></b>	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.
<b>Low Time</b>	The time the signal remains below the Base reference level. The measurement is taken on each cycle in the waveform record.
<b>Negative Duty Cycle<sup>2</sup></b>	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.
<b>Negative Pulse Width<sup>2</sup></b>	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.
<b>Period<sup>2</sup></b>	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.

<sup>2</sup> This measurement can also be taken on digital signals.

Measurement	Description
<b>Phase<sup>2</sup></b>	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.
<b>Positive Duty Cycle<sup>2</sup></b>	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.
<b>Positive Pulse Width<sup>2</sup></b>	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.
<b>Rise Time</b>	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.
<b>Rising Slew Rate</b>	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.
<b>Setup Time<sup>2</sup></b>	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.
<b>Skew<sup>2</sup></b>	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.
<b>Time Outside Level</b>	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.
<b>Unit Interval<sup>2</sup></b>	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 228

**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, the jitter measurements are moved to the **Jitter** tab of the **Add Measurements** menu. See [The Jitter tab \(Advanced Jitter and Eye Analysis\) \(option 5-DJA, 6-DJA\)](#) on page 217.

---

**Jitter Measurements panel measurements.**

Measurement	Description
<b>Phase Noise</b>	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.
<b>TIE</b>	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 228

**The Jitter tab (Advanced Jitter and Eye Analysis) (option 5-DJA, 6-DJA)**

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 (optional).

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 (optional).**

Measurement	Description
<b>DCD</b>	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.
<b>DDJ</b>	Data dependent jitter. The peak-to-peak amplitude for the component of the deterministic jitter correlated with the data pattern in the waveform. The measurement is taken on the entire record.
<b>DJ</b>	Deterministic jitter. The peak-to-peak amplitude of all timing errors that exhibit deterministic behavior. The measurement is taken across the entire record.
<b>DJ-<math>\delta\delta</math></b>	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.
<b>F/2</b>	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by two. The measurement is taken on the entire record.
<b>F/4</b>	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by four. The measurement is taken across the entire record.
<b>F/8</b>	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by eight. The measurement is taken on the entire record.
<b>J2</b>	The total jitter at a bit error rate of 2.5e-3 (TJ@2.5e-3). The measurement is taken on the entire record.
<b>J9</b>	The total jitter at a bit error rate of 2.5e-3 (TJ@2.5e-3). The measurement is taken on the entire record.
<b>Jitter Summary</b>	Adds multiple jitter measurements to the measurement badge, and displays Bathtub, TIE Spectrum trend plot, TIE Histogram, Eye Diagram plots on the screen.
<b>NPJ</b>	Nonperiodic jitter. The portion of the BUJ (Bounded uncorrelated jitter) that is random. BUJ excludes DDJ, DCD, and RJ. The measurement is taken on the entire record.
<b>Phase Noise</b>	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.
<b>PJ</b>	Periodic jitter. The peak-to-peak amplitude of the uncorrelated sinusoidal components of the deterministic jitter. The measurement is taken on the entire record.
<b>RJ</b>	Random jitter. The RMS magnitude of all random timing errors following a Gaussian distribution. The measurement is taken on the entire record.
<b>RJ-<math>\delta\delta</math></b>	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.

Measurement	Description
<b>SRJ</b>	Sub-rate jitter. The composite jitter due to periodic components at $\frac{1}{2}$ , $\frac{1}{4}$ , and $\frac{1}{8}$ of the data rate. The measurement is taken on the entire record.
<b>TIE</b>	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.
<b>TJ@BER</b>	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 (optional).**

Measurement	Description
<b>Eye Height</b>	The minimum vertical eye opening at the center of the recovered unit interval. The measurement is taken on the entire waveform record.
<b>Eye High</b>	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.
<b>Eye Low</b>	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.
<b>Eye Width</b>	The minimum horizontal eye opening at the center of the recovered unit interval. The measurement is taken on the entire waveform record.
<b>EyeHeight@BER</b>	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.
<b>EyeWidth@BER</b>	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.
<b>Q-Factor</b>	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 (optional).**

Measurement	Description
<b>AC Common Mode (Pk-Pk)</b>	The peak-to-peak of the common mode voltage of the two specified sources. The measurement is taken on the entire record.
<b>Bit Amplitude</b>	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).
<b>Bit High</b>	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).
<b>Bit Low</b>	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).
<b>DC Common Mode</b>	The arithmetic mean of the common mode voltage of two sources. The measurement is taken on the entire record.
<b>Differential Crossover</b>	The voltage level of a differential signal pair at the crossover point(s). The measurement is taken at each crossover point in the record.
<b>T/nT Ratio</b>	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 (optional).**

Measurement	Description
<b>SSC Freq Dev</b>	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.
<b>SSC Modulation Rate</b>	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 228

### The Power tab (optional)

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.

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 Measurements panel (optional)* on page 222

*Amplitude Analysis Measurements panel (optional)* on page 223

*Timing Analysis Measurements panel (optional)* on page 223

*Switching Analysis Measurements panel (optional)* on page 224

*Output Analysis Measurements panel (optional)* on page 225

*Magnetic Analysis Measurements panel (optional)* on page 224

*Frequency Response Analysis Measurements panel (optional)* on page 225

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 (optional).

Measurement	Description
<b>Power Quality</b>	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.
<b>Harmonics</b>	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.
<b>Input Capacitance</b>	Measures the DUT input capacitance when powered on.
<b>Inrush Current</b>	Measures the positive and negative peak input current during DUT power-on.

---

**NOTE.** You should take the Input Capacitance and Inrush Current measurements independently. Input Capacitance and Inrush Current measurements use **Power Preset** to optimize the settings and trigger to measure the DUT power-on signal, which may result in inaccurate measurements for other measurement types.

---



**Amplitude Analysis Measurements panel (optional).**

Measurement	Description
<b>Cycle Amplitude</b>	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.
<b>Cycle Top</b>	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.
<b>Cycle Base</b>	The most common data value below the midpoint of the waveform. This measurement is made on each cycle in the record.
<b>Cycle Peak-to-Peak</b>	The difference between the Maximum and Minimum values in the measurement region. This measurement is made on each cycle in the record.
<b>Cycle Maximum</b>	The maximum data point. This measurement is made on each cycle in the record.
<b>Cycle Minimum</b>	The minimum data point. This measurement is made on each cycle in the record.

**Timing Analysis Measurements panel (optional).**

Measurement	Description
<b>Period</b>	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.
<b>Frequency</b>	Frequency is the reciprocal of Period ( $\text{Frequency} = 1/\text{Period}$ ). This measurement is made on each cycle in the record.
<b>Positive Duty Cycle</b>	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.
<b>Negative Duty Cycle</b>	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.
<b>Positive Pulse Width</b>	The time the signal remains above the Mid-reference level. This measurement is taken on each cycle of the waveform record or measurement region.
<b>Negative Pulse Width</b>	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 (optional).**

Measurement	Description
<b>Switching Loss</b>	The mean instantaneous power and energy in the turn-on, turn-off, and conduction regions of a switching device. Provides the SWL trajectory plot.
<b>dv/dt</b>	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 ( $R_B$ ).
<b>di/dt</b>	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 ( $R_B$ ).
<b>SOA</b>	An X-Y plot of switching device voltage and current. The SOA mask testing provides pass/fail testing to component specifications.
<b><math>R_{DS(on)}</math></b>	The resistance (the slope of the V-I curve) when the switching device is conducting.

**Magnetic Analysis Measurements panel (optional).**

Measurement	Description
<b>Inductance</b>	The integral of the voltage divided by the current of a magnetic component during circuit operation.
<b>Magnetic Property</b>	The B-H curve for a magnetic component during circuit operation.
<b>Magnetic Loss</b>	The average value of the product of the voltage and current through the inductor. This represents the total loss of the magnetic device and consists of resistive and eddy current losses during circuit operation.
<b>I vs. <math>\int V</math></b>	The integral of voltage against current.

**Output Analysis Measurements panel (optional).**

Measurement	Description
Line Ripple	The RMS and peak-to-peak values of the line frequency portion of the AC signal.
Switching Ripple	The RMS and peak-to-peak values of the input signal.
Efficiency	The ratio of output power to input power for a power conversion circuit.
Turn On Time	Measures the time from when the input voltage is applied to the DUT to the time when the output voltage reaches a steady level.
Turn Off Time	Measures the time from when the input voltage is removed from the DUT to the time when the output voltage reaches a zero level.

---

**NOTE.** You should take the Turn On Time and Turn Off Time measurements independently. Turn On Time and Turn Off Time measurements use the **Power Preset** function to optimize the settings and trigger to measure the DUT power-on signal, which may result in inaccurate measurements for other measurement types.

---

**NOTE.** If you save a session file that includes power measurements, make sure to recall (load) the session file into an instrument with the same number of channels that was used to create the session file.

---

**Frequency Response Analysis Measurements panel (optional).**

Measurement	Description
Control Loop Response	Plots the gain as $20 \log (V_{out}/V_{in})$ and phase difference between $V_{in}$ and $V_{out}$ at each frequency within the swept band. The resulting plot is commonly referred to as a Bode Plot.
Power Supply Rejection Ratio	Measures both the modulated input and output AC voltage levels and plots the rejection ratio as $20 \log (V_{in}/V_{out})$ at each frequency within the specified band.

**See also.** [Power measurement configuration menu overview \(optional\)](#) on page 242

**The DDR measurements tab (Requires option SUP6-DBDDR3)**

The DDR tab lists the double data rate (DDR) measurements that you can add to the Results bar. DDR measurements include amplitude and timing analysis. The DDR tab is shown only if you have purchased and installed the SUP6-DBDDR3 DDR Analysis option.

To open the DDR Measurements tab and add a measurement to the Results bar:

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

To add a measurement to the Results bar:

1. Select the signal source/s.
2. Tap a measurement panel:
  - Amplitude Analysis measurement panel*
  - Timing Analysis measurement panel*
3. Select a 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.

**Amplitude Measurements panel (Requires option SUP6-DBDDR3).**

Measurement	Description
<b>AOS</b>	The total area of the signal above the specified reference level.
<b>AUS</b>	The total area of the signal below the specified reference level.
<b>Vix(ac)</b>	The differential input cross-point voltage measured from the actual crossover voltage and its complement signal to a designated reference voltage. This is measured on a single ended signal.  <i><b>NOTE.</b> This measurement requires option SUP6-DJA.</i>
<b>AOS Per tCK</b>	The total area of the signal that crosses the specified reference level calculated over consecutive periods. It is applicable to clock and address/command waveforms only.
<b>AUS Per tCK</b>	The total area of the signal that crosses the specified reference level calculated over consecutive periods. It is applicable to clock and address/command waveforms only.
<b>AOS Per UI</b>	The total area of the signal that crosses the specified reference level calculated over consecutive unit intervals. It is applicable to data and data strobe waveforms only.
<b>AUS Per UI</b>	The total area of the signal that crosses the specified reference level calculated over consecutive unit intervals. It is applicable to data and data strobe waveforms only.

**Timing Measurements panel (Requires option SUP6-DBDDR3).**

Measurement	Description
tRPRE	The width of the Read burst preamble. This is measured from the exit of tristate to the first driving edge of the differential strobe.
tWPRE	The width of the Write burst preamble. It is measured from the exit of tristate to the first driving edge of the differential strobe.
tPST	The width of Read or Write burst' postamble. It is measured from the last falling edge crossing the mid reference level to the start of an undriven state (as measured by a rising trend per JEDEC specification).
Hold Diff	The elapsed time between the designated edge of the single ended waveform and the designated edge of a differential waveform.  The measurement uses the closest single ended waveform edge to the differential waveform edge that falls within the range limits.
Setup Diff	The elapsed time between the designated edge of a single ended waveform and when the differential waveform crosses its own voltage reference level.  The measurement uses the closest single ended waveform edge to the differential waveform edge that falls within the range limits.
tCH(avg)	The average high pulse width calculated across a sliding 200 cycle window of consecutive high pulses.
tCK(avg)	The average clock period across a sliding 200-cycle window.
tCL(avg)	The average low pulse width calculated across a sliding 200 cycle window of consecutive low pulses.
tCH(abs)	The high pulse width of the differential clock signal. It is the amount of time the waveform remains above the mid reference voltage level.
tCL(abs)	The low pulse width of the differential clock signal. It is the amount of time the waveform remains below the mid reference voltage level.
tJIT(duty)	The largest elapsed time between tCH and tCH(avg) or tCL and tCL(avg) for a 200-cycle window.
tJIT(per)	The largest elapsed time between tCK and tCK(avg) for a 200-cycle sliding window.
tJIT(cc)	The absolute difference in clock period between two consecutive clock cycles.
tERR(n)	The cumulative error across multiple consecutive cycles from tCK(avg). It measures time difference between the sum of clock period for a 200-cycle window to n times tCK(avg).
tERR(m-n)	The cumulative error across multiple consecutive cycles from tCK(avg). It measures the time difference between the sum of clock periods for a 200-cycle window to n times tCK(avg).

Measurement	Description
tDQSCK	The strobe output access time from the differential clock. It is measured between the rising edge of the clock before or after the differential strobe Read preamble time. The edge locations are determined by the mid-reference voltage levels.
tCMD-CMD	The elapsed time between two logic states.
tCKSRE	The valid clock cycles required after Self Refresh Entry (SRE) command. Changing the input clock frequency or the supply voltage is permissible only after tCKSRE time when the SRE command is registered.
tCKSRX	The valid clock cycles required before the Self Refresh Exit (SRX) command. Changing the input clock frequency or the supply voltage is permissible provided the new clock frequency or supply voltage is stable for the tCKSRX time prior to SRX command.

**See also.** [DDR measurement configuration menu overview \(Requires option SUP6-DBDDR3\)](#) on page 269

## 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
<b>Measurement Statistics</b> (Measurement name panel)	A list of measurement statistics related to the measurement. You can add these statistics to a measurement badge by selecting the <b>Show Statistics in Badge</b> control.
<b>Show Statistics in Badge</b> (Measurement name panel)	Adds the listed statistical measurement readouts to the measurement badge readout.
<b>Plots</b> (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 <a href="#">Add Plot configuration menu</a> on page 306.
<b>Configure panel</b>	Sets the source, label text, and other fields that are specific to each measurement type.  See <a href="#">Configure panel (Measurement configuration menu)</a> on page 230.
<b>Reference Levels panel</b>	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 <a href="#">Reference Levels panel (Measurement configuration menu)</a> on page 231.
<b>Clock Recovery panel</b> (jitter measurements)	Sets the clock recovery settings for some jitter measurements.  See <a href="#">Clock Recovery panel (Measurement configuration menu)</a> on page 234.
<b>Gating panel</b>	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 <a href="#">Gating panel (Measurement configuration menu)</a> on page 240.
<b>Filter/Limit Results panel</b>	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 <a href="#">Filter/Limit Results panel (Measurement Settings menu)</a> on page 241.

**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
<b>Measurement Statistics</b>	A list of measurement statistics. You can add these statistics to a measurement badge by selecting the <b>Show Statistics in Badge</b> control.
<b>Show Statistics in Badge</b>	Select to add the listed statistical measurement readouts to the measurement badge.
<b>Plots</b>	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 230

[Reference Levels panel \(Measurement configuration menu\)](#) on page 231

[Clock Recovery panel \(Measurement configuration menu\)](#) on page 234

[Gating panel \(Measurement configuration menu\)](#) on page 240

[Filter/Limit Results panel \(Measurement Settings menu\)](#) on page 241

**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
<b>Source</b>	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.
<b>Label</b>	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.



Field or control	Description
<b>Signal Type</b>	Sets the signal type (Clock, Data, Auto) of the source signal for some measurements.
<b>Edge, Clock edge</b>	Sets the edge of the signal to use for starting the measurement.
<b>From Edge</b>	Sets the Source 1 waveform edge on which to start the measurement, for two-source measurements.
<b>To Edge</b>	Sets the Source 2 waveform edge on which to stop the measurement, for two-source measurements.
<b>Calculate One Measurement Per</b>	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.
<b>Pattern Detection</b>	Auto attempts to detect the type of pattern and set
<b>Pattern Type</b>	Sets whether the source signal data is a Repeating pattern or an Arbitrary pattern type.
<b>Pattern Length</b>	Sets the pattern length to use when Pattern Type = Repeating.
<b>Window Length</b>	Sets the overall window length to use when Pattern Type = Arbitrary.

#### Other measurement panels.

*Measurement Name panel (Measurement configuration menu)* on page 230

*Reference Levels panel (Measurement configuration menu)* on page 231

*Clock Recovery panel (Measurement configuration menu)* on page 234

*Clock Recovery- Advanced Settings configuration menu* on page 239

*Gating panel (Measurement configuration menu)* on page 240

*Filter/Limit Results panel (Measurement Settings menu)* on page 241

#### 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
<b>Reference Levels</b>	<p>Global sets whether the reference levels defined in this measurement apply to all measurements that are set to global (the default setting).</p> <p>Local sets the Reference Level parameters to apply to just this measurement.</p>
<b>Source</b>	Lists the source signals used for each edge of the measurement.
<b>Set Levels In</b>	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.
<b>Levels</b>	<p>Sets the reference levels as specified percentages of the Top and Base waveform measurement.</p> <p>To set custom reference values, tap <b>Custom</b>, tap a setting field, and use the Multipurpose Knob to set the different % (relative) or absolute values.</p> <p><b>High</b> and <b>Low</b> references are used to calculate rise and fall times. The default High reference is 90% and Low reference is 10%.</p> <p><b>Mid</b> reference is primarily used for measurements between edges such as pulse widths. The default level is 50%.</p>

Field or control	Description
<b>Base Top Method</b>	<p>Sets the method to calculate the waveform Base and Top values, which is then used to calculate the High, Mid, and Low reference levels.</p> <p><b>Auto</b> is the default method, and automatically determines the best Base Top method to use. Most commonly sets the Top Base method to Histogram Mode.</p> <p><b>MinMax</b> 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.</p> <p><b>Histogram Mean</b> 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.</p> <p><b>Histogram Mode</b> 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.</p> <p><b>Histogram Eye Center</b> 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.</p> <hr/> <p><b>NOTE.</b> <i>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.</i></p>
<b>Hysteresis</b>	<p>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.</p>

#### Other measurement settings panels.

[Measurement Name panel \(Measurement configuration menu\)](#) on page 230

[Configure panel \(Measurement configuration menu\)](#) on page 230

[Clock Recovery panel \(Measurement configuration menu\)](#) on page 234

[Clock Recovery- Advanced Settings configuration menu](#) on page 239

[Gating panel \(Measurement configuration menu\)](#) on page 240

[Filter/Limit Results panel \(Measurement Settings menu\)](#) on page 241

**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
<b>Clock Recovery</b>	<p><b>Global</b> 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).</p> <p><b>Local</b> sets the Clock Recovery parameters to apply to just this measurement.</p>
<b>Method</b>	<p><b>PLL</b> simulates the behavior of the specified hardware Phase Locked Loop clock recovery circuit to derive the clock signal.</p> <p><b>Constant Clock</b> uses linear regression so that the recovered clock minimizes the mean squared sum of the Time Interval Error (TIE) for that waveform.</p> <p><b>Explicit Clock</b> derives the reference clock from a specified channel other than the one upon which the measurement is defined.</p>
<b>Standard</b>	<p>Sets the standard to use for the PLL model. Information for a selected standard is listed under the drop-down list.</p> <p>Only available when Method = PLL.</p>
<b>Mode</b>	<p><b>Mean</b> chooses both the frequency and the phase to minimize the mean squared error. Available when Method = Constant Clock.</p> <p><b>Median</b> chooses the phase so that the median error between the recovered and measured edges is zero. Only available when Method = Constant Clock.</p> <p><b>Fixed</b> 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.</p> <p><b>Select Explicit Clock-Edge</b> 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.</p> <p><b>Select Explicit Clock-PLL</b> 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.</p>

Field or control	Description
<b>Calculate On</b>	<p><b>First Acq.</b> 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.</p> <p><b>Every Acq.</b> the clock-recovery algorithm to choose a new best-fit clock frequency and phase for each new oscilloscope acquisition.</p> <p>Clearing the measurement results will reset the clock recovery so that both frequency and phase are optimized on the subsequent acquisition.</p> <p>Only available when Method = Constant Clock and Mode = Mean or Median.</p>
<b>Clock Frequency</b>	<p>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.</p> <hr/> <p><b>NOTE.</b> <i>This method typically results in a closed eye.</i></p> <hr/> <p>Only available when Method = Constant Clock and Mode = Fixed.</p>
<b>Loop BW</b>	<p>Sets the PLL loop bandwidth. Displays the Closed Loop bandwidth that has been configured based on the current standard.</p> <p>Only available when Method = PLL.</p>
<b>Clock Source</b>	<p>Sets the source for the explicit clock.</p> <p>Only available when Method = Explicit Clock.</p>
<b>Clock Edge</b>	<p>Set whether the rising, falling or both edges of the clock source should be considered.</p> <p>Only available when Method = Explicit Clock.</p>
<b>Clock Multiplier</b>	<p>Set the number of edges to be used.</p> <p>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.</p> <p>Only available when Method = Explicit Clock.</p>
<b>Clock Offset</b>	<p>Set to Auto or Manual.</p>

Field or control	Description
<b>Offset</b>	<p>Sets the clock offset amount relative to data.</p> <p>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.</p> <p>Only available when Clock Offset = Manual.</p>
<b>PLL Model</b>	<p>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.</p> <p>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.</p> <p>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/s<sup>2</sup> 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).</p> <p>Only available when Method = Explicit Clock and Mode = Explicit Clock - PLL.</p>
<b>JTF BW</b>	<p>Displays or sets the Jitter Transfer Function bandwidth that has been configured based on the current standard.</p> <p>Only available when Method = Explicit Clock and Mode = Explicit Clock - PLL and PLL Model = Type II.</p>
<b>Damping</b>	<p>Sets the damping factor for the PLL. It is enabled only for Type II phase-locked loop.</p> <p>Only available when Method = Explicit Clock and Mode = Explicit Clock - PLL and PLL Model = Type II.</p>
<b>Advanced</b>	<p>Opens the Clock Recovery-Advanced Settings configuration menu to refine the recovered clock signal. See <a href="#">Clock Recovery- Advanced Settings configuration menu</a> on page 239.</p> <p>Only available when Method = PLL.</p>

**About constant clock recovery.** In Constant Clock Recovery, the clock is assumed to be of the form  $A \cdot \sin(2\pi ft + \text{phase})$ , 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 230

*Configure panel (Measurement configuration menu)* on page 230

*Reference Levels panel (Measurement configuration menu)* on page 231

*Clock Recovery- Advanced Settings configuration menu* on page 239

*Gating panel (Measurement configuration menu)* on page 240

*Filter/Limit Results panel (Measurement Settings menu)* on page 241



### 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
<b>Advanced Clock Recovery Method</b>	Sets the method used to recover a clock from the measured signal. Available selections are <b>None</b> (default), <b>Nominal Data Rate</b> and <b>Known Data Pattern</b> .
<b>Bit Rate</b>	Sets the clock bit rate. The <b>Bit Rate</b> field is only present when <b>Advanced Clock Recovery Method</b> is set to <b>Nominal Data Rate</b> .
<b>Pattern File</b>	Drop-down list with 20 most recent pattern files loaded onto the oscilloscope.  Available when Advanced Clock Recovery = Known Data Pattern.
<b>Browse</b>	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 230

[Configure panel \(Measurement configuration menu\)](#) on page 230

[Reference Levels panel \(Measurement configuration menu\)](#) on page 231

[Clock Recovery panel \(Measurement configuration menu\)](#) on page 234

[Gating panel \(Measurement configuration menu\)](#) on page 240

[Filter/Limit Results panel \(Measurement Settings menu\)](#) on page 241

**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
<b>Gating</b>	Sets whether this measurement's gating settings are <b>Global</b> or <b>Local</b> . When <b>Global</b> is selected, changing anything in this panel causes the same change to all other measurements that also have <b>Global</b> selected.  When <b>Local</b> is selected, settings in this panel only effect this measurement.
<b>Gating Type</b>	Sets the gate type used to take measurements.  <b>None:</b> Measurements are taken across the entire record.  <b>Screen:</b> 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.  <b>Cursors:</b> Measurements are taken on the portion of the waveform between the cursors.  <b>Logic:</b> Measurements are taken only when the logical state of a specified waveform is true.
<b>Source</b>	Sets the source to use for Logic or Search gates.  If Gating Type = <b>Logic</b> , the Source field lists all analog channels, math waveforms and reference waveforms.  If Gating Type = <b>Search</b> , the Source field lists all defined searches.
<b>Threshold</b>	Sets the threshold value for the Logic gate source to be considered a logic 1 value.
<b>Hysteresis</b>	Sets the Hysteresis value for the Logic gate source.
<b>Active</b>	Sets the logic state value for the Logic gate source.

**See also.**

[Measurement Name panel \(Measurement configuration menu\)](#) on page 230

[Configure panel \(Measurement configuration menu\)](#) on page 230

[Reference Levels panel \(Measurement configuration menu\)](#) on page 231

[Clock Recovery panel \(Measurement configuration menu\)](#) on page 234

[Clock Recovery- Advanced Settings configuration menu](#) on page 239

[Filter/Limit Results panel \(Measurement Settings menu\)](#) on page 241

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

**Other measurement settings panels.**

*Measurement Name panel (Measurement configuration menu)* on page 230

*Configure panel (Measurement configuration menu)* on page 230

*Reference Levels panel (Measurement configuration menu)* on page 231

*Clock Recovery panel (Measurement configuration menu)* on page 234

*Clock Recovery- Advanced Settings configuration menu* on page 239

*Gating panel (Measurement configuration menu)* on page 240

## Power measurement configuration menu overview (optional)

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
<b>Measurement Statistics</b> (Measurement name panel)	A list of measurement statistics related to the measurement. You can add these to a measurement badge by selecting the <b>Show Statistics in Badge</b> control.
<b>Show Statistics in Badge</b> (Measurement name panel)	Adds the listed statistical measurement readouts to the measurement badge readout.
<b>Plots</b> (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, B-H curve, I vs. $\int V$ , Inductance, Power Supply Rejection Ratio, Control Loop Response, and Instantaneous Math.  To add a plot to the screen, tap the plot button. See <a href="#">Add Plot configuration menu</a> on page 306.

Field, control, or panel	Description
<b>Configure panel</b>	Sets the source, label text, and other fields that are specific to each measurement type.  See <a href="#">Configure panel (Measurement configuration menu)</a>
<b>Reference Levels panel</b>	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 <a href="#">Reference Levels panel (Measurement configuration menu)</a> on page 231
<b>Gating panel</b>	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 <a href="#">Gating panel (Measurement configuration menu)</a> on page 240

### 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.

---

**NOTE.** *The Power Measurement Name panel and the Configure panel are merged together for Input Capacitance, Inrush Current, Turn On Time, Turn Off Time, Control Loop Response, and Power Supply Rejection Ratio power measurements.*

---

Field or control	Description
<b>Power Autoset</b>	Sets the oscilloscope acquisition system for optimal results for all active power measurements except Inrush Current, Input Capacitance, Turn-on Time, and Turn-off time. See <a href="#">Power Autoset</a> .
<b>Power Preset</b>	Sets the oscilloscope acquisition system for optimal results for Turn On Time, Turn Off Time, Input Capacitance, and Inrush Current power measurements. See <a href="#">Power Preset</a> on page 262.  Available only for Turn On Time, Turn Off Time, Input Capacitance, Inrush Current, Power Supply Rejection Ratio, and Control Loop Response power measurements.
<b>Measurement Statistics</b>	Shows a list of measurement statistics. You can add these statistics to a measurement badge by selecting the <b>Show Statistics in Badge</b> control.
<b>Show Statistics in Badge</b>	Adds the listed statistical measurement readouts to the measurement badge.
<b>Plots</b>	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, Control Loop Response, PSRR, and SOA.  Trend adds the trend plot to the Waveform view.

**See also.**

[Configure panel \(Measurement configuration menu\)](#) on page 230

[Reference Levels panel \(Measurement configuration menu\)](#) on page 231

[Gating panel \(Measurement configuration menu\)](#) on page 240

**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 and dv/dt measurement: Configure panel.**

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

### Harmonics measurement: Configure panel.

Field or control	Description
<b>Standard</b>	<p>Sets the standard to use for measurements. None (no Standard), IEC 61000-3-2, MIL-STD-1399, AM14, or D0-160G (Standard for Airborne equipment. Supported for harmonics measurement for single phase DUT).</p> <p><b>NOTE.</b> When standard is set as None with the current source, it displays an error as Low Amplitude. Change relevels method to MinMax manually. This error is displayed when you recall the demo setup files for this measurement.</p>
<b>Harmonics</b>	Sets the harmonics order (number of harmonics) for the selected standard. Ranges from 40 to 100.
<b>Voltage Source</b>	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
<b>Current Source</b>	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
<b>Line Frequency</b>	<p>Sets the line frequency of the power signal source.</p> <p>In Auto mode, the application measures the frequency of the input signal automatically. Input signal is used to compute harmonics in the current source. Default is Auto.</p>
<b>Label</b>	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.
<b>Harmonics Source</b>	Selects the source of harmonics calculation source (Voltage or Current) for computation of the Harmonics.
<b>Power Level (MIL-STD-1399)</b>	Selects the required power level, High or Low. It is used to compute limit values for MIL-STD-1399.
<b>Current (MIL-STD-1399)</b>	Selects either rated or measured. Specify the value for rated or measure the Input current signal.
<b>Start Frequency</b>	<p>Specify the fundamental frequency to be measured for each standard.</p> <ul style="list-style-type: none"> <li>■ <b>None:</b> The values are <b>Auto</b>, 50, 60, 400, or <b>Custom</b> (specify the start Frequency)</li> <li>■ <b>IEC-61000-3-2:</b> The values are <b>Auto</b>, 50, 60, or <b>Custom</b></li> <li>■ <b>MIL-STD-1399:</b> The values are <b>Auto</b>, 400, or <b>Custom</b></li> <li>■ <b>AM-14:</b> The values are <b>Auto</b>, 50, 60, or <b>Custom</b></li> <li>■ <b>D0-160G:</b> The values are <b>Auto</b>, 360, 400, 800, or <b>Custom</b></li> </ul> <p>Default is <b>Auto</b>. The measured frequency value is displayed in the measurement badge.</p>

**Power Quality measurement: Configure panel.**

Field or control	Description
<b>Voltage Source</b>	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
<b>Current Source</b>	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
<b>Label</b>	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.
<b>Calculate Over Full Cycles</b>	Enables calculating the measurement over the entire acquisition.
<b>Frequency Reference</b>	Selects the signal source (Voltage or Current) to use to determine the measurement frequency.

**Input Capacitance measurement: Configure panel.**

Field or control	Description
<b>Voltage Source</b>	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
<b>Current Source</b>	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
<b>Label</b>	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.
<b>Peak Voltage</b>	Sets the peak voltage value of the DUT. <b>Power Preset</b> uses the specified peak voltage value to compute the vertical scale.
<b>Peak Current</b>	Sets the peak current value of the DUT. <b>Power Preset</b> uses the specified peak current value to compute the vertical scale.

**Inrush Current measurement: Configure panel.**

Field or control	Description
<b>Current Source</b>	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
<b>Label</b>	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.
<b>Peak Current</b>	Sets the peak current value of the DUT. <b>Power Preset</b> uses the specified peak current value to compute the vertical scale.



**SOA measurement: Configure panel.**

Field or control	Description
<b>Voltage Source</b>	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
<b>Current Source</b>	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
<b>Label</b>	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.
<b>Define Mask</b>	Defines the linear mask for SOA measurements. See <a href="#">Define Mask</a>

**Switching Loss measurement: Configure panel.**

Field or control	Description
<b>Voltage Source</b>	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
<b>Current Source</b>	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
<b>Label</b>	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.
<b>Type</b>	<p>Select SMPS / PFC/ Flyback:</p> <ul style="list-style-type: none"> <li>■ <b>SMPS</b>: 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.</li> <li>■ <b>PFC</b>: Select this option when input DUT signals are from Power Factor Correction Circuit. For this case, Vg source is mandatory.</li> <li>■ <b>Flyback</b>: Select this option when input signals are ringing. This option does not require a Vg source.</li> </ul> <hr/> <p><b>NOTE.</b></p> <ul style="list-style-type: none"> <li>■ <i>For SMPS and PFC settings to correctly calculate results, set REF Level to "Units" instead of "Percentage."</i></li> <li>■ <i>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.</i></li> </ul>
<b>Gate Voltage (Vg)</b>	<p>Sets the Vg input source, which is a clean signal.</p> <p>Available when Type = <b>SMPS</b> or <b>PFC</b>.</p>
<b>Vg Level Ton - Start</b>	<p>Selects the source of computation of the harmonic. In the voltage source, standard is always <b>None</b>.</p> <p>Available when Gate Voltage (Vg) ≠ <b>None</b>.</p>
<b>PWM Type</b>	Select <b>Fixed</b> or <b>Variable</b> based on the varying pulse width of the switching signal.
<b>Conduction calculation</b>	Select <b>MOSFET</b> or <b>BJT/IGBT</b> semiconductor types.
<b>R<sub>DS(on)</sub></b>	<p>If <b>MOSFET</b> is selected, then R<sub>DS(on)</sub> is used to compute for conduction Loss.</p> <p>Available when Conduction Calculation = <b>MOSFET</b></p>
<b>V<sub>CE(sat)</sub></b>	<p>If <b>BJT/IGBT</b> is selected, then V<sub>CE(sat)</sub> is used to compute for conduction loss.</p> <p>Available when Conduction Calculation = <b>BJT/IGBT</b>.</p>

Field or control	Description
<b>Set On/Off Levels In:</b>	Sets the REF levels for computation of the $T_{ON}$ and $T_{OFF}$ regions. Levels can be set in % or absolute values.
<b><math>T_{off}</math>-Stop Current Level</b>	Sets the $T_{on}$ -Start and $T_{off}$ -Stop of the max switch current.
<b><math>T_{on}</math>-Stop &amp; <math>T_{off}</math> Start Voltage Level</b>	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 <b>Set On/Off Levels In:</b> control.

#### Inductance measurement: Configure panel.

Field or control	Description
<b>Voltage Source</b>	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
<b>Current Source</b>	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
<b>Label</b>	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.
<b>Edge Source</b>	Selects the voltage or current source for computing edges.

**NOTE.** When you are measuring the inductance of a transformer, do not load the secondary winding. The measurement of the inductance at the primary winding under no load condition is as good as measuring the inductance for a single winding. When you are measuring the inductance of the coupled inductor with multiple windings on the same core, the measured value of the inductance will deviate from the actual value due to the influence of the current on other windings. You can use this measured value to calculate the Ripple current.

#### Magnetic Loss and I vs. $\int V$ measurement: Configure panel.

Field or control	Description
<b>Voltage Source</b>	Selects the voltage source used to take the measurement. Tap the field to show the list of available sources.
<b>Current Source</b>	Selects the current source used to take the measurement. Tap the field to show the list of available sources.
<b>Label</b>	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.

**Efficiency measurement: Configure panel.**

Field or control	Description
<b>Voltage Source</b>	Selects the input voltage source used to take the measurement. Tap the field to show the list of available sources.
<b>Current Source</b>	Selects the input current source used to take the measurement. Tap the field to show the list of available sources.
<b>Label</b>	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.
<b>No of Outputs (1, 2, or 3)</b>	Sets the number of outputs on the SMPS DUT, up to a maximum of three. You can configure these outputs in the application, and set up the sources (V and I), so that the efficiency per output and total efficiency for all available outputs are displayed.
<b>Output 1 Voltage</b>	Selects the voltage source for output one when the number of outputs is set to one.
<b>Output 1 Current</b>	Selects the current source for output one when the number of outputs is set to one.

**Turn On Time measurement: Configure panel.**

Field or control	Description
<b>Input Source</b>	Selects the channel connected to the input side of the DUT.
<b>Type</b>	Selects the input to output power conversion type. Default is <b>DC-DC</b> .
<b>Label</b>	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.
<b>Maximum Voltage</b>	Sets the maximum input voltage, in the range of 1 V to 500 V.
<b>Input Trigger</b>	Sets the input trigger level, in the range of 1 V to 500 V.
<b>Frequency</b>	Sets the DUT input AC signal frequency, in the range of 1 Hz to 1 MHz.  Available when <b>Type = AC-DC</b>
<b>Maximum Time</b>	Sets the maximum time window in which to capture the turn on time measurement (from the initial state to a steady state), in the range of 1 $\mu$ s to 500 s, with a resolution of 0.1 $\mu$ s.
<b>Number of Outputs (1, 2, 3, 4, 5, 6, or 7)</b>	Sets the number of outputs to measure on the DUT (maximum of seven).
<b>Output Source</b>	Selects the channel source to use for the listed output.
<b>Output Voltage</b>	Sets the expected maximum voltage for each listed output, in the range of -6 kV to +6 kV, with a resolution of 0.001 V. The measurement uses this value to set the vertical scale units for each output waveform.  The instrument measures the actual maximum voltage on each output, and uses this value to determine the output On/Off state levels, where On is $\geq 90\%$ of the measured value, and Off is $\leq 10\%$ of the measured value.

**NOTE.** The configuration value for maximum voltage and trigger level will not be remembered when it changes from AC-AC, AC-DC, DC-AC, and DC-DC modes. It displays the default values.

**Turn Off Time measurement: Configure panel.**

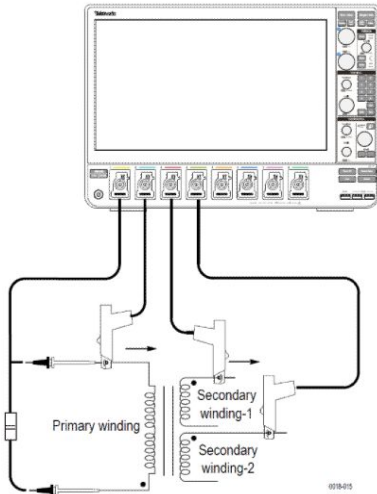
Field or control	Description
<b>Input Source</b>	Selects the channel connected to the input side of the DUT.
<b>Type</b>	Selects the input to output power conversion type. Default is <b>AC-DC</b> .
<b>Label</b>	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
<b>Maximum Voltage</b>	Specify the maximum input voltage, in the range of 1 V to 500 V.
<b>Input Trigger</b>	Specify the input trigger level, in the range of 1 V to 500 V.
<b>Frequency</b>	Specify the DUT input AC signal frequency, in the range of 1 Hz to 1 MHz.  Available when <b>Type = AC-DC</b>
<b>Maximum Time</b>	Sets the maximum time window to capture the turn off time measurement (from the initial state to a steady state), in the range of 1 $\mu$ s to 5 s, with a resolution of 0.1 $\mu$ s.
<b>Number of Outputs (1, 2, 3, 4, 5, 6, or 7)</b>	Sets the number of outputs to measure on the DUT (maximum of seven).
<b>Output Source</b>	Selects the channel source to use for the listed output.
<b>Output Voltage</b>	Sets the expected maximum voltage for each listed output, in the range of -6 kV to +6 kV, with a resolution of 0.001 V. The measurement uses this value to set the vertical scale units for each output waveform.  The instrument measures the actual maximum voltage on each output, and uses this value to determine the output On/Off state, where $\geq 90\%$ of the measured value = On, and $\leq 10\%$ of the measured value = Off.

**$R_{DS(on)}$  measurement: Configure panel.**

Field or control	Description
<b>Voltage Source</b>	Selects the source used to take the voltage measurement. Tap the field to show the list of available sources.
<b>Current Source</b>	Selects the source used to take the current measurement. Tap the field to show the list of available sources.
<b>Label</b>	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.
<b>Device Type</b>	Sets the semiconductor (transistor device) type.  <b>Switching (v/i)</b> is the ratio of voltage to current in Ohms.  <b>PN Junction / Diode (dv/di)</b> is the ratio of rate of change of voltage to current in Ohms.

**Magnetic Property measurement: Configure panel.**

Field or control	Description
<b>Primary Voltage Source</b>	Selects the source used to take the voltage measurement. Tap the field to show the list of available sources.
<b>Primary Current Source</b>	Selects the source used to take the current measurement. Tap the field to show the list of available sources.
<b>Primary Turns</b>	Sets the number of turns used on the primary winding. Tap the field and use the knob to set the value, or double-tap on the field and use the keyboard to enter a value.
<b>Label</b>	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.
<b>Edge Source</b>	<p>Sets the source used to detect the edge. Select either <b>Voltage</b> or <b>Current</b>. Default is <b>Voltage</b>.</p> <hr/> <p><b>NOTE.</b> For a variable switching operation, connect the gated drive signal to the edge source, as in variable mode the amplitude of the voltage acquired across the inductor varies with time. It is recommended to use gate drive signal as it is clean with no variations.</p> <hr/>
<b>Units</b>	<p>Sets the measurement units.</p> <p><b>SI</b> sets the dimensions of the component in meters and units of magnetics in Tesla and Amperes Turns per meter.</p> <p><b>CGS</b> sets the unit of measurement to centimeters, and the result units are in Gauss and Oersted.</p>
<b>Cross Section</b>	<p>Sets the cross section dimensions of the magnetic component. The values are:</p> <ul style="list-style-type: none"> <li>■ 1 nm<sup>2</sup> to 1 Mm<sup>2</sup> for <b>SI</b></li> <li>■ 1 ncm<sup>2</sup> to 1 Mcm<sup>2</sup> for <b>CGS</b></li> </ul>
<b>Magnetic Length</b>	<p>Sets the cross section dimensions of the magnetic length. The values are:</p> <ul style="list-style-type: none"> <li>■ 0 m to 1 Mm for <b>SI</b></li> <li>■ 0 cm to 1 Mcm for <b>CGS</b></li> </ul>

Field or control	Description
<p><b>Number of sec windings</b></p>	<p>Sets the number of secondary windings to measure. The range is from 1 to 6. Measure the magnetic property of secondary windings such as in a coupled inductor or a transformer that has multiple windings on the same core.</p> <p>Follow the connection combinations of the voltage and current probe at the primary or the main winding at the secondary or the other windings to the · dot as shown in the following figure. The voltage probe should be connected such that the voltage is read as positive when the current rises.</p>  <p><b>WARNING.</b> When connecting to a circuit with hazardous voltages, see the warnings for the individual products and verify that the probes and other components used are within their ratings. For more information, refer to the topic General safety summary.</p>
<p><b>Winding (1-6)</b></p>	<p>Sets the signal source used to measure the selected winding. The range is from 1 to 6.</p>
<p><b>Turns (1-6)</b></p>	<p>Sets the number of turns in the respective secondary winding.</p>



**Amplitude Analysis, Timing Analysis power measurements: Configure panel.**

Field or control	Description
<b>Source</b>	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.
<b>Label</b>	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.
<b>Edge</b>	Sets the signal edge to detect (rise or fall). Used in most <b>Timing Analysis</b> measurements.  Available when the power measurement = <b>Frequency</b> .
<b>Clock Edge</b>	Sets the clock signal edge to detect (rise, fall, or either). Used in <b>Timing Analysis</b> measurements.  Available when the power measurement = <b>Positive Duty Cycle</b> or <b>Negative Duty Cycle</b> .
<b>Line Frequency</b>	Sets the line frequency of the power signal source. <b>Power Autoset</b> uses the selected frequency to setup the scope acquisition parameters.  Available when the power measurement = <b>Line Ripple</b> .
<b>Switching Frequency</b>	Specify the operating frequency of the switching device. <b>Power Autoset</b> uses the selected frequency to set the oscilloscope acquisition parameters. Frequency range is 50 Hz to 1 MHz.  Only available when the power measurement = <b>Switching Ripple</b> .

**Control Loop Response measurement: Configure panel.** See [Connection setup for Frequency Response Analysis \(FRA\) measurements](#) on page 258.

Field or control	Description
<b>Generator</b>	<p>Generates stimulus, as a series of sine waves with configured amplitude, which are send to the input of the DUT.</p> <p>Supported generator is Internal signal source.</p> <p>The maximum value is 50, the minimum is 10 and the default value is 20 points.</p>
<b>Impedance</b>	User configurable Impedance for the generator.
<b>Points Per Decade</b>	<p>Sets the number of frequency points between the start and stop frequency in terms of log scale.</p> <p>The minimum value is 10, maximum value is 100, and the default is 10.</p>
<b>Start/Stop Frequency</b>	Sets the generator start and stop frequency. The maximum stop frequency depends on the generator. The start frequency is 100 Hz, the stop frequency is 20 MHz, and the default is 100 Hz.
<b>Amplitude Mode</b>	<p>Sets the built-in signal generator output amplitude mode.</p> <p><b>Constant:</b> Sets the constant amplitude to all frequencies in the in-built signal generator.</p> <p><b>Profile:</b> Opens a configuration menu that lets you set the amplitude for each frequency band.</p> <p>The generator uses the values in the table to set the output amplitude as the frequency changes.</p>
<b>Configure Profile</b>	<p>Opens <b>Profile</b> configuration menu to enter the frequency range with amplitude values. See <a href="#">Configure Profile</a> on page 260.</p> <p>Available only when Amplitude mode = <b>Profile</b>.</p>
<b>Amplitude</b>	<p>Sets the generator to apply the same amplitude to all frequencies output from the generator.</p> <p>The minimum value is 20 mV as per internal generator, the maximum value is whatever the generator supports, and the default is 100 mV.</p> <p>Available when Amplitude mode = <b>Constant</b>.</p>

---

**NOTE.** *Math sources are not supported for Control Loop Response (BODE) and PSRR measurements, so do not configure Math for these measurements.*

---

**Power Supply Rejection Ratio (PSRR) measurement: Configure panel.** See [Connection setup for Frequency Response Analysis \(FRA\) measurements](#) on page 258.

Field or control	Description
<b>Generator</b>	<p>Generates stimulus, as a series of sine waves with configured amplitude, which are send to the input of the DUT.</p> <p>Supported generator is Internal signal source.</p> <p>The maximum value is 50, the minimum is 10 and the default value is 20 points.</p>
<b>Impedance</b>	Sets the output impedance of the built-in signal generator
<b>Points Per Decade</b>	<p>Sets the number of frequency points between the start and stop frequency in terms of log scale.</p> <p>The minimum value is 10, maximum value is 100, and the default is 10.</p>
<b>Start/Stop Frequency</b>	Sets the generator start and stop frequency. The maximum stop frequency depends on the built-in signal generator. The start frequency is 100 Hz, the stop frequency is 20 MHz, and the default is 100 Hz.
<b>Amplitude Mode</b>	<p>Sets the built-in signal generator output amplitude mode.</p> <p><b>Constant:</b> Sets the constant amplitude to all frequencies in the built-in signal generator.</p> <p><b>Profile:</b> Opens a configuration menu that lets you set the amplitude for each frequency band. The generator uses the values in the table to set the output amplitude as the frequency changes.</p>
<b>Configure Profile</b>	<p>Opens <b>Profile</b> configuration menu to enter the frequency range with amplitude values. See <a href="#">Configure Profile</a> on page 260.</p> <p>Available only when Amplitude mode = <b>Profile</b>.</p>
<b>Amplitude</b>	<p>Sets the generator to apply the same amplitude to all frequencies output from the generator.</p> <p>The minimum value is 20 mV as per internal generator, the maximum value is whatever the generator supports, and the default is 100 mV.</p> <p>Available when Amplitude mode = <b>Constant</b>.</p>
<b>Power Supply</b>	Displays the power supply used. You cannot control the power supply programmatically from the application.
<b>Voltage</b>	Sets the voltage value manually in the power supply.
<b>Current</b>	Sets the current value manually in the power supply.

**Other measurement configuration panels.**

*Reference Levels panel (Measurement configuration menu)* on page 231

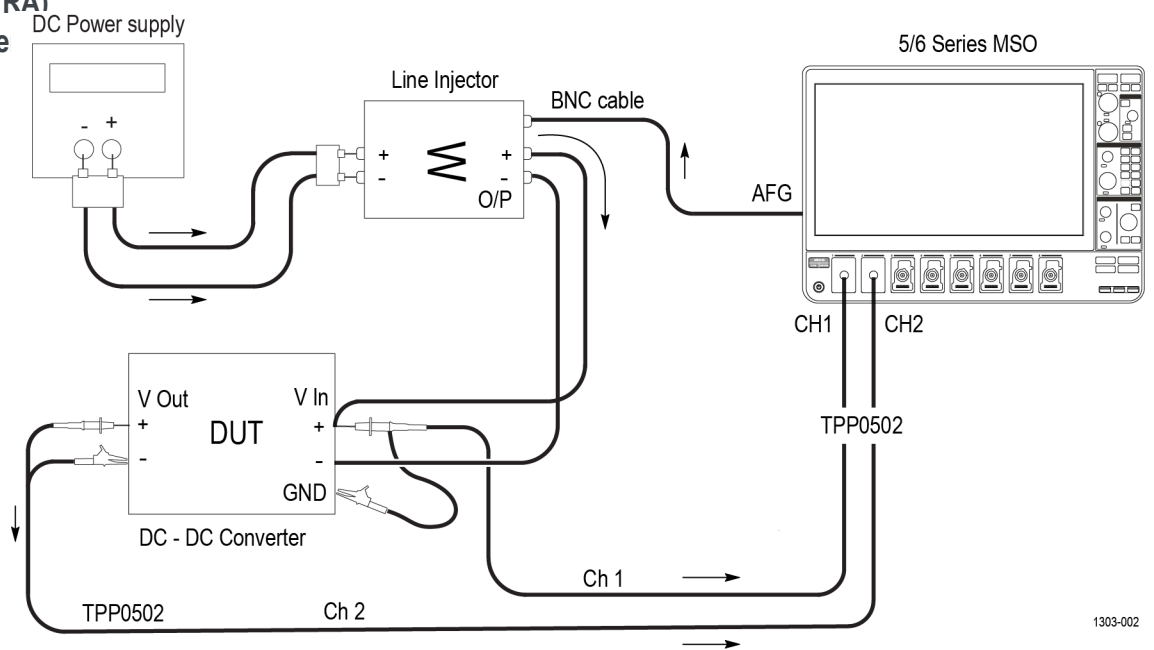
*Power Measurement Name panel (Measurement configuration menu)* on page 243

*Configure panel (Measurement configuration menu)* on page 230

*Gating panel (Power measurement configuration menu)* on page 268

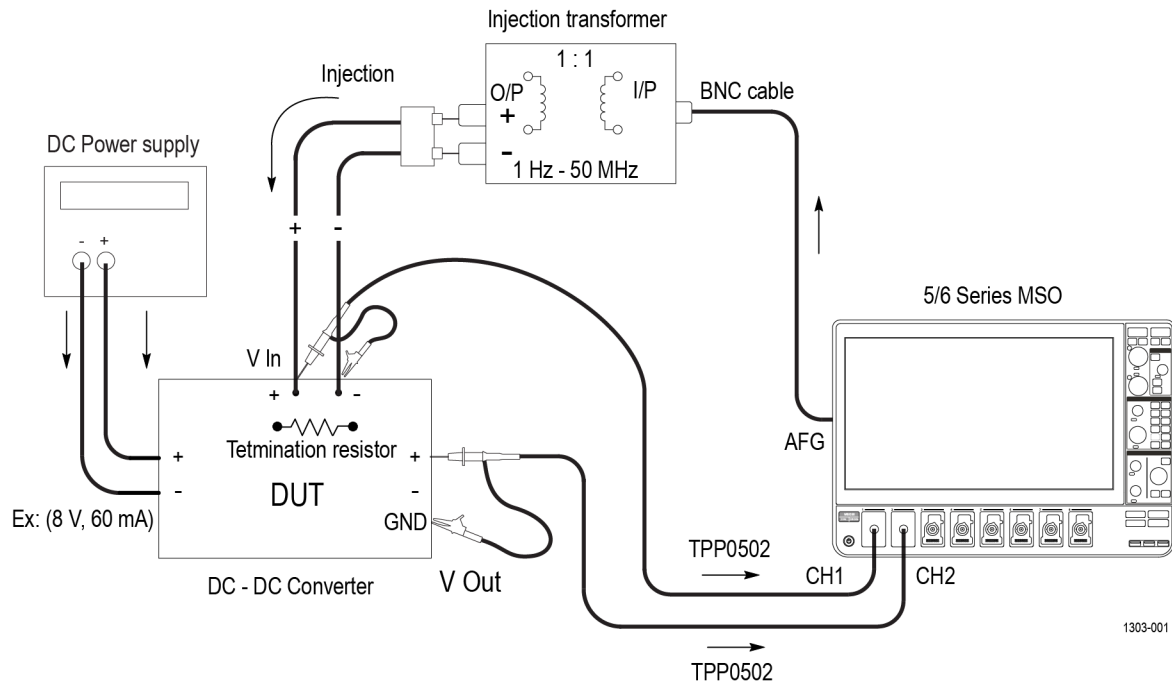
**Connection setup for Frequency Response Analysis (FRA) measurements**

Frequency Response Analysis (FRA) measurements include Control Loop Response (Bode) and Power Supply Rejection Ratio (PSRR) measurements.



**Figure 15: Connection diagram for Bode**

1303-002



**Figure 16: Connection diagram for PSRR**

Use the following steps to make the FRA measurements:

1. Connect the output of the built-in AFG of the oscilloscope to the DUT through a transformer.  
This generates the stimulus which are sequence of sine waves with the configured amplitudes. The stimulus is send to the DUT.  
Transformer used between the built-in generator and the DUT input, should be a Line Injector (PSRR)/ Injection Transformer (Bode).
2. This generates the stimulus, and it is send to the DUT as its input.
3. Connect the DUT using external power supply like Keithley or any proprietary power supply.
4. Connect the probing from oscilloscope channel 1 to the  $V_{in}(+/-)$  of the DUT.
5. Connect the probing from oscilloscope channel 2 to the  $V_{out}(+/-)$  of the DUT.
6. Power on the DUT.

**Configure Profile**

Use the Configure Profile menu to configure the amplitude profile for the Power Supply Rejection Ratio (PSRR) and Control Loop Response measurements.

It is recommended to customize the built-in signal generator amplitude values using the amplitude profile configuration for these measurements. With profiling you can set the generator at lower amplitudes, at different frequencies where the DUT is sensitive, and set higher amplitudes where the DUT is less sensitive to distortion. In general, a Switched-mode Power Supply (SMPS) is very sensitive near the zero degree cross over frequency, so it is advisable to define a profile, instead of using a constant amplitude output signal.

Use the parameters to configure the profile for PSRR and Control Loop Response measurements.

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

Use the following steps to configure the amplitude profile for a PSRR and Control Loop Response measurement:

1. Click **Configure Profile** button. Displays the configure profile dialog. By default the configure profile dialog displays two rows:
  - The Start Frequency of the first row is the Start Frequency from the PSRR measurement configure panel.
  - The Stop Frequency of the first row is the mean of Start and Stop Frequency values from the PSRR/Control Loop Response measurement configure panel.
  - The Start Frequency of the second row is 'greater' than the Stop Frequency from the PSRR/Control Loop Response measurement configure panel.
  - The Stop Frequency of the second row is the Stop Frequency from the PSRR/Control Loop Response measurement configure panel.
2. You can change the values, to reduce the variations in the gain curve at particular frequencies where the DUT is sensitive to distortion. This can be done by making configuration changes and viewing the response in the gain/phase curve of the Control Loop Response/PSRR plot.
3. Click anywhere outside the configure profile table to account for the updates.

Below are the sample amplitude values for a DC-DC convertor DUT.

**Table 7: Sample Amplitude values**

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

---

**NOTE.** The stop frequency can be configured up to 50 MHz which is maximum value of build-in AFG.

---

**Power Autose**

Sets the oscilloscope acquisition system for optimal measurement results for all active power measurements except Turn On Time, Turn Off Time, Input Capacitance, and Inrush Current.

Use the following steps to run a **Power Autose** on applicable power measurements:

1. Add the power measurements that you want to take to the Results bar.
2. Configure each power measurement individually (input voltage sources, current sources, label name, and so on).
3. Connect the input signals to the instrument and confirm the waveforms are correct and not clipping.
4. Double-tap the power measurement result badge for which you want to take a measurement.

5. Tap the **Power Autose** button for the measurement and wait for the busy indicator to disappear. The instrument has now been optimized for that power measurement.

---

**NOTE.**

- *In case of failure, the instrument displays a popup error message. See [Errors and Warnings](#).*
  - *When different frequency signals are connected to different channels, **Power Autose** uses the AC signal of the lowest-numbered channel to evaluate and set up the instrument parameters. For example, If Channel 1 is connected to a DC signal, Channel 2 is connected to an AC signal of 1 MHz, and Channel 4 is connected to an AC signal of 1 kHz, then **Power Autose** uses Channel 2 (the first channel with an AC signal) to set up the instrument parameters. Channel 4 may not be set up to see the minimum number of cycles properly; you may need to manually set the Channel 4 parameters.*
  - ***Power Autose** runs all parts of instrument Autose, even if some parts have been disabled in the **User Preferences > Autose** panel. See [Autose panel fields and controls](#) on page 422 for more information on the Autose function.*
  - ***Power Autose** is available per measurement. You should run the Power Autose for each measurement separately when you add measurements, to ensure optimum power measurement setup for that measurement.*
- 

6. Record the power measurement after the **Power Autose** completes.
7. Repeat steps 4 - 6 for other Power measurements that you want to take.

### Power Preset

Sets the oscilloscope vertical, horizontal and acquisition parameters using the user inputs like peak voltage and current for Input Capacitance, Inrush Current, Turn On Time, and Turn Off Time. For Control Loop response and PSRR measurements, it sets the built-in generator such as start frequency, impedance, and amplitude values and also sets the vertical and horizontal acquisition parameters, so that oscilloscope is ready to acquire the desired waveform once the measurement starts.

---

**NOTE.** *The **Power Preset** button is available only for Input Capacitance, Inrush Current, Turn On Time, Turn Off Time, Control Loop Response, and Power Supply Rejection Ratio power measurements.*

---

Use the following steps to run a **Power Preset** on applicable power measurements:

1. Add the supported power measurements to the **Results** bar.
2. Double-tap each Measurement badge and configure the measurement (input voltage sources, current sources, label name, and so on).
3. Connect the input signals.
4. Tap the **Power Preset** button in the measurement configuration to set.



To set the vertical and horizontal acquisition parameters:

- a. Measurement configuration begins and displays a window (displayed only for FRA measurements).

---

**NOTE.**

- *When the measurement runs, you cannot operate oscilloscope front panel buttons.*
  - *You can change the ACQ modes to Sample mode after **Power Preset** is clicked and before oscilloscope **RUN/STOP** is performed*
- 

- b. To stop the measurement configuration or to operate the front panel buttons, click the **Abort** button on the window.
- 

**NOTE.** *When the FRA measurement is configured, the window disappears.*

---

- c. Tap **Run/Stop** on the front panel of the oscilloscope.
5. Follow the instructions on the Measurement badge to take the power measurement.

### SOA Mask definition controls and fields

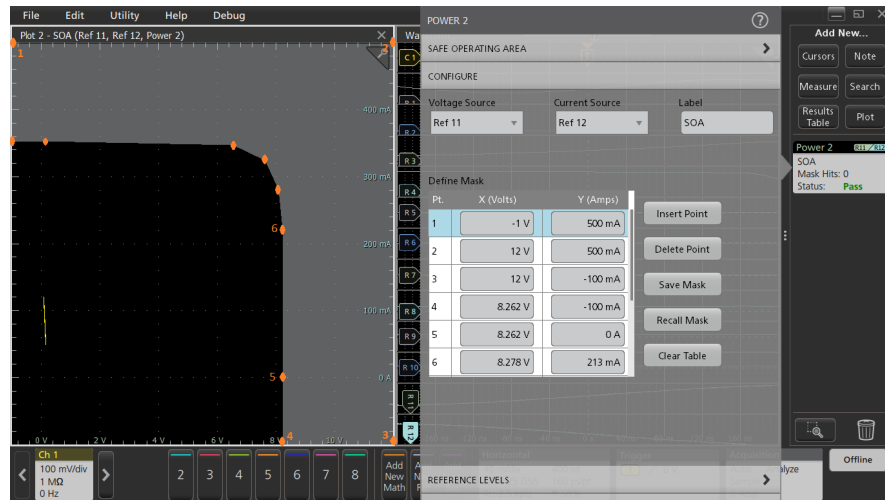
Use the SOA Mask dialog to configure the parameters to add point, delete point, save mask, and recall mask.

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

#### Define Mask fields and controls

Field or control	Description
<b>X (Volts)</b>	Define voltage values for the mask point.
<b>Y (Amps)</b>	Define current values for the mask point.
<b>Insert Point</b>	Add voltage and current points to define mask. Points are added to the end of the existing list.
<b>Delete Point</b>	Deletes the selected point data row.
<b>Save Mask</b>	Opens the <b>Save As</b> menu to navigate to and select the location at which to save the SOA mask data as a .pwrmsk file.
<b>Recall Mask</b>	Opens the <b>Open</b> menu to navigate to and select the location from which to recall (load) the SOA mask data .pwrmsk file.  <b>NOTE.</b> <i>Mask files created with oscilloscope firmware version 1.4.x cannot be used in firmware version 1.6.x. Use version 1.6.x and later to create and recall mask files.</i>
<b>Clear Table</b>	Clears the values of the mask coordinates in the table.

The following image shows the default SOA mask and its associated mask point table.



**NOTE.** You need to define both internal and external points.

### 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
<b>Save Location</b>	<p>Lists the location where the file will be saved. The default value is the last location to which a file was saved.</p> <p>Use the <b>Browse</b> 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.</p> <p>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.</p>
<b>Browse</b>	<p>Tap to open the Browse Save As Location configuration menu, to navigate to the location to which to save the file. See <a href="#">Browse Save As Location configuration menu</a> on page 414.</p>
<b>File Name</b>	<p>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.</p> <p>Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names.</p> <p>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.</p>
<b>Save As Type</b>	<p>You can only save SOA mask files as type .pwrmsk.</p>
<b>Cancel</b>	<p>Cancels the file save action and closes the configuration menu. You can also cancel the save operation by tapping anywhere outside the menu.</p>
<b>Save</b>	<p>Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.</p>

**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.

---

**NOTE.** Mask files created with oscilloscope firmware version 1.4.x cannot be used in firmware version 1.6.x. Use version 1.6.x and later to create and recall mask files.

---

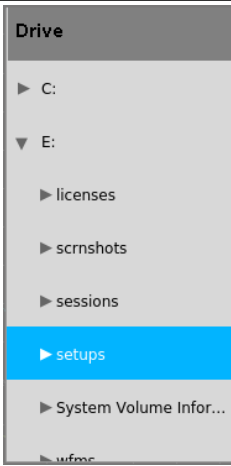


To open the Recall Mask configuration menu:

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 Microsoft Windows 10 Operating System SSD.** Instruments with 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 Windows 10 installed), which assign a non-changing drive letter to each USB port.

**Recall file configuration menu (SOA mask file).**

Field or control	Description
<b>Look in:</b>	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.
<b>File name</b>	Lists the selected file name to recall. Tap on the file name in the Name column to add it to this field.
<b>Files of type</b>	The SOA mask file type (.pwrmsk) cannot be changed.
<b>Cancel</b>	Cancels the file recall action and closes the menu.
<b>Recall</b>	Recalls the selected SOA mask file and plots it on the SOA Plot view.

**USB port drive names and locations.** Use the following 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
<b>MSO54, MSO56, MSO58, MSO64, without Windows OS</b>		
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)
<b>MSO58LP, without Windows OS</b>		
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)
<b>Instruments with Windows OS and USB port labels</b>		
Root drive	C	User-accessible memory on the oscilloscope.
USB ports	<b>Dynamic port letter assignment</b>	If Windows operating system is installed, 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 231 for the Reference panel fields and controls.

**Other measurement settings panels.** *Power Measurement Name panel (Measurement configuration menu)* on page 243

*Configure panel (Power measurement configuration menu)* on page 244

*Gating panel (Power measurement configuration menu)* on page 268

**Gating panel (Power  
measurement  
configuration menu)**

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

See *Gating panel (Measurement configuration menu)* on page 240 for the Gating panel fields and controls.

**See also**

*Power Measurement Name panel (Measurement configuration menu)* on page 243

*Reference Levels panel (Power measurement configuration Menu)* on page 268

*Configure panel (Power measurement configuration menu)* on page 244

## DDR measurement configuration menu overview (Requires option SUP6-DBDDR3)

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

To open the DDR measurement configuration menu for a measurement, double-tap a DDR 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). The measurement name panel only shows fields and controls relevant to the selected measurement. Common controls include displaying statistics in the measurement badge and creating plots of the measurement.

The most common DDR measurement name fields are listed in the following table.

### DDR Measurement configuration menu fields, controls, and panels

Field, control, or panel	Description
<b>Show Statistics in Badge</b> (Measurement name panel)	Adds the listed statistical measurement readouts to the measurement badge readout.
<b>Plots</b> (Measurement name panel)	Buttons that open Plot views of the measurement. Available plots depend on the measurement. Plot types include Time Trend, Histogram, and Spectrum.  To add a plot to the screen, tap the plot button.  See <a href="#">Add Plot configuration menu</a> on page 306.
<b>Panels</b>	
<b>Configure panel</b>	Sets the source, label text, and other fields that are specific to each measurement type.  See <a href="#">Configure panel (DDR measurement configuration menu)</a> on page 271.
<b>Reference Levels panel</b>	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 <a href="#">Reference Levels panel (DDR measurement configuration Menu)</a> on page 271.

Field, control, or panel	Description
<b>Gating panel</b>	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 <a href="#">Gating panel (DDR measurement configuration menu)</a> on page 272.
<b>Filter/Limit Results panel</b>	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 <a href="#">Filter/Limit results panel (DDR measurement configuration Menu)</a> on page 272.

**DDR measurement name panel (Measurement configuration menu)**

The DDR 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 and configure a DDR measurement, double-tap a DDR measurement badge.

The contents of the measurement configuration menu depend on the measurement.

Field or control	Description
<b>Show Statistics in Badge</b>	Adds the listed statistical measurement readouts to the measurement badge.
<b>Plots</b>	Adds a Plot view of the measurement to the screen. Available plots depend on the measurement. Plot types include <b>Time Trend</b> , <b>Histogram</b> , and <b>Spectrum</b> .

**See also.**

[Configure panel \(DDR measurement configuration menu\)](#) on page 271

[Filter/Limit results panel \(DDR measurement configuration Menu\)](#) on page 272

[Reference Levels panel \(DDR measurement configuration Menu\)](#) on page 271

[Gating panel \(DDR measurement configuration menu\)](#) on page 272



**Configure panel (DDR 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 DDR measurement:

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

**Configure panel.**

Field or control	Description
<b>Source</b>	Sets the signal source used to take the measurement. Tap the field to show the list of available sources. Select the current source.
<b>Label</b>	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

**Other measurement configuration panels.**

[Filter/Limit results panel \(DDR measurement configuration Menu\)](#) on page 272

[Reference Levels panel \(DDR measurement configuration Menu\)](#) on page 271

[Gating panel \(DDR measurement configuration menu\)](#) on page 272

**Reference Levels panel (DDR 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 231 for the Reference panel fields and controls.

**Other measurement settings panels**

[Configure panel \(DDR measurement configuration menu\)](#) on page 271

[Filter/Limit results panel \(DDR measurement configuration Menu\)](#) on page 272

[Gating panel \(DDR measurement configuration menu\)](#) on page 272

**Gating panel (DDR measurement configuration menu)**

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

See *Gating panel (Measurement configuration menu)* on page 240 for the Gating panel fields and controls.

**See also**

*Configure panel (DDR measurement configuration menu)* on page 271

*Filter/Limit results panel (DDR measurement configuration Menu)* on page 272

*Reference Levels panel (DDR measurement configuration Menu)* on page 271

**Filter/Limit results panel (DDR measurement configuration 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).

See *Filter/Limit Results panel (Measurement Settings menu)* on page 241 for the Filter/Limit Results panel fields and controls.

**Other measurement settings panels**

*Configure panel (DDR measurement configuration menu)* on page 271

*Reference Levels panel (DDR measurement configuration Menu)* on page 271

*Gating panel (DDR measurement configuration menu)* on page 272

## 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
<b>Display</b>	Toggles bus display on or off.
<b>Label</b>	Enter label text in this field. The default label is the name of the bus type.
<b>Position</b>	Sets the vertical position of the bus waveform. Default is 0 (center of graticule).
<b>Set to 0</b>	Sets the vertical position of the bus waveform to 0 (center graticule).
<b>Bus Type</b>	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 <a href="#">Serial bus and trigger options</a> on page 19.
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.
<b>Display format</b>	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.
<b>Decode format</b>	Sets how decoded data information is shown in the bus. Select from listed formats. Available formats depend on the bus type.

**Bus type configuration menus**

[ARINC 429 serial bus menu](#) on page 274

[Audio serial bus configuration menu](#) on page 275

[CAN serial bus configuration menu](#) on page 277

[Ethernet serial bus menu](#) on page 280

[FlexRay serial bus configuration menu](#) on page 282

[I2C serial bus configuration menu](#) on page 284

[LIN serial bus configuration menu](#) on page 288

[MIL-STD-1553 serial bus menu](#) on page 290

[Parallel Bus configuration menu](#) on page 292

[RS232 serial bus menu](#) on page 295

[SENT serial bus configuration menu](#) on page 297

[SPI serial bus configuration menu](#) on page 299

[SPMI serial bus configuration menu](#) on page 301

[USB serial bus configuration menu](#) on page 303

**ARINC 429 serial bus menu**

Use the ARINC 429 bus menu (optional) to set up and decode a ARINC 429 avionics network serial data bus waveform.

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
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>ARINC429</b> .
<b>Polarity</b>	Select the polarity to match the ARINC 429 bus being acquired.
<b>Source</b>	Select the ARINC 429 signal source.
<b>High Threshold, Low Threshold</b>	Sets the valid high and low threshold values for the signal source.
<b>Bit Rate</b>	Sets the bit rate to 12,500, 100,000, or Custom.
<b>Custom Rate</b>	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.
<b>Data Format</b>	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).
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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 275

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

**Audio serial bus configuration menu**

Use the Audio bus menu (optional) to set up and display Audio Type I2S, Left Justified (LJ), Right Justified (RJ), or TDM Audio serial bus waveforms.

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
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>Audio</b> .
<b>Audio Type</b>	Sets the digital audio signal type. Select from the drop-down list.
<b>Bit Order</b>	Set the waveform to decode with most-significant (MS) bit first or least-significant (LS) bit first.
<b>Bit Clock</b>	Set the signal source, logic level threshold, and polarity for the Bit Clock signal.
<b>Word Select</b>	Set the signal source, logic level threshold, and normal or invert signal setting for the Word signal.
<b>Data</b>	Set the signal source, logic level threshold, and logic definition (active high or low) for the Data signal.
<b>Word Size</b> (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).
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	Sets the decode format used to display the bus information.
<b>TDM-specific settings</b>	
<b>Frame Sync</b>	Set the signal source, logic level threshold, and polarity for the frame sync signal.
<b>Data Bits per Channel</b>	Set the number of data bits per audio channel.
<b>Clock Bits per Channel</b>	Set the number of clock bits per audio channel.
<b>Channels per Frame</b>	Set the number of audio channels per data frame.
<b>Bit Delay</b>	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 274

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*FlexRay serial bus configuration menu* on page 282

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

**CAN serial bus configuration menu**

Use the CAN bus menu (optional) to set up and display a CAN (Controller Area Network) or CAN FD (CAN Flexible Datarate) serial bus waveform.

To create a new CAN bus waveform:

1. Tap **Add New Bus** on the Settings bar.
2. Double-tap the CAN badge to open the bus configuration menu.
3. Set the **Bus Type** to **CAN**.

To change the settings on an existing CAN serial bus waveform, double-tap the CAN waveform badge on the Settings bar to open the configuration menu.

**CAN serial bus menu fields and controls.**

Field or control	Description
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>CAN</b> to set up and display a CAN bus waveform.
<b>Signal Type</b>	Sets the CAN signal type to decode. The default is <b>CAN_H</b> .
<b>CAN Standard</b>	Sets the CAN signal standard to decode. The default is <b>CAN 2.0</b> .
<b>Source</b>	Select the signal source from listed analog and digital channels.
<b>Threshold</b>	Sets the high/low logic transition level.
<b>Sample Point</b>	Sets the sample point from 5% to 95% of the position within the bit period or the unit interval.
<b>Bit Rate</b>	Sets the bit rate of your CAN bus serial data.  To enter a custom bit rate, select <b>Custom</b> and enter the custom bit rate in the <b>Custom Rate</b> input box.  Only available when <b>CAN Standard = CAN 2.0</b> .
<b>SD Bit Rate</b>	Sets the SD bit rate of your CAN FD serial bus data.  To enter a custom bit rate, select <b>Custom</b> and enter the custom bit rate in the <b>Custom Rate</b> input box.  Only available when <b>CAN Standard = CAN FD (ISO) or CAN FD (non-ISO)</b> .
<b>FD Bit Rate</b>	Sets the FD bit rate of your CAN FD serial bus data.  To enter a custom bit rate, select <b>Custom</b> and enter the custom bit rate in the <b>Custom Rate</b> input box.  Only available when <b>CAN Standard = CAN FD (ISO) or CAN FD (non-ISO)</b> .
<b>Custom rate</b>	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 <b>Bit Rate or SDI Bit Rate = Custom</b> .



Field or control	Description
<b>Display Format</b>	<p><b>Bus</b> sets the Waveform view to show just the decoded bus information.</p> <p><b>Bus and Waveform</b> sets the Waveform view to show both the decoded bus and the source signal waveforms.</p> <p>You can also tap on the + symbol on the bus waveform to toggle between showing the bus only or showing bus and source waveforms.</p>
<b>Decode Format</b>	<p>Sets the decode format used to display the bus information. Formats are <b>Hex</b>, <b>Binary</b>, <b>Mixed Hex</b>, and <b>Symbolic</b>.</p> <p>Selecting <b>Symbolic</b> lets you navigate to and load a dbc decode label file for the current CAN waveform. The .dbc file is a proprietary description of a CAN bus system and its message and signal labels. Loading the .dbc file displays the message and signal information on the decoded CAN waveform. Selecting a .dbc file immediately loads the .dbc contents into the instrument.</p> <hr/> <p><b>NOTE.</b> <i>.dbc files are not saved as part of a save session operation.</i></p> <hr/> <p>To remove the .dbc symbolic information from the waveform, select a different <b>Decode Format</b>.</p>
<b>Browse</b>	<p>Opens the standard <b>File Open</b> dialog to navigate to and load a .dbc file to apply to the bus being created. The <b>File Open</b> dialog automatically filters for the .dbc file extension.</p> <p>Only available when <b>Decode Format = Symbolic</b>.</p>
<b>Path field</b>	<p>Use to enter the path to a .dbc file, or select from a drop-down list of recently accessed .dbc files.</p> <p>Only available when <b>Decode Format = Symbolic</b>.</p>

**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 274

*Audio serial bus configuration menu* on page 275

*Ethernet serial bus menu* on page 280

*FlexRay serial bus configuration menu* on page 282

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

**Ethernet serial bus menu**

Use the Ethernet bus menu (optional) to set up and display an Ethernet 10BaseT or 100BaseT serial bus waveform.

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
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>Ethernet</b> .
<b>Speed</b>	Sets the network signal speed.
<b>Signal Type</b>	Sets the signal type to Single Ended or Differential.
<b>Source</b>	Sets the signal source for Differential. Only available when Signal Type = Diff.
<b>D+ Input, D- Input</b>	Defines the signal sources and threshold values for the Single Ended data+ and - signals.  Only available when Signal Type = Single Ended.
<b>High Threshold</b>	Sets the threshold value for a logic 1 value.  Only available when Signal Type = Diff.
<b>Low Threshold</b>	Sets the threshold value for a logic 0 value.  Only available when Signal Type = Diff.
<b>IPv4</b>	Set to Yes if the Ethernet signal being measured uses Internet Protocol version 4 (IPv4).
<b>Q-(VLAN)</b>	Set to Yes if the Ethernet signal being measured uses IEEE 802.1Q virtual LANs.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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 274

*Audio serial bus configuration menu* on page 275

*CAN serial bus configuration menu* on page 277

*FlexRay serial bus configuration menu* on page 282

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

### **FlexRay serial bus configuration menu**

Use the Flexray bus menu (optional) to set up and display a Flexray automotive network serial bus waveform.

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
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>FlexRay</b> .
<b>Signal Type</b>	Select the FlexRay signal type being measured.
<b>Channel Type</b>	Set to A or B channel.
<b>Source</b>	Select the FlexRay signal source.
<b>Threshold</b>	Sets the threshold value for the TX or RX signal type.
<b>High Threshold, Low Threshold</b>	Sets the high and low threshold values for the BM Inverted and Bdiff/BP signal types.
<b>Bit Rate</b>	Select a bit rate. To set a custom bit rate, select Custom and enter a value in the Custom Rate field.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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 274

*Audio serial bus configuration menu* on page 275

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

### **I2C serial bus configuration menu**

Use the I2C bus menu (optional) to set up and display an I<sup>2</sup>C (Inter-Integrated Circuit) serial bus waveform.

To set up the I<sup>2</sup>C serial bus menu:

- To create a new I<sup>2</sup>C 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<sup>2</sup>C serial bus waveform, double-tap the I<sup>2</sup>C **Bus** waveform badge and make necessary changes in the configuration menu.

**I2C serial bus menu fields and controls.**

Field or control	Description
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>I2C</b> .
<b>SCLK Input</b>	Sets the source and threshold level for the Serial Clock Line signal.
<b>SDA Input</b>	Sets the source and threshold level for the Serial Data signal.
<b>Include R/W bit in Address</b>	Select <b>Yes</b> 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 <b>No</b> to display 7-bit addresses as seven bits, and 10-bit addresses as ten bits.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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 274

*Audio serial bus configuration menu* on page 275

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*FlexRay serial bus configuration menu* on page 282

*I3C serial bus configuration menu* on page 286

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

**I3C serial bus configuration menu**

Use the I3C bus menu (optional) to set up and display a decoded I<sup>3</sup>C serial bus waveform.

To set up the I<sup>3</sup>C serial bus menu:

- To create a new I<sup>3</sup>C 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 **I3C**.
- To change the settings on an existing I<sup>3</sup>C serial bus waveform, double-tap the I<sup>3</sup>C **Bus** waveform badge and make necessary changes in the configuration menu.



**I3C serial bus menu fields and controls.**

Field or control	Description
<b>Display</b>	Turns <b>On</b> or <b>Off</b> displaying the bus on the Waveform view.
<b>Label</b>	Creates 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.
<b>Position</b>	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in <b>Stacked</b> mode), or center screen in <b>Overlay</b> mode. The unit of position is screen divisions.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>I3C</b> .
<b>SCLK Input</b>	Sets the source and threshold level for the Serial Clock Line signal.
<b>SDA Input</b>	Sets the source and threshold level for the Serial Data signal.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	Sets the decode format used to display the bus information. Formats are <b>Hex</b> , <b>Binary</b> , and <b>Mixed Hex</b> .

**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 274

[Audio serial bus configuration menu](#) on page 275

[CAN serial bus configuration menu](#) on page 277

[Ethernet serial bus menu](#) on page 280

[FlexRay serial bus configuration menu](#) on page 282

[I2C serial bus configuration menu](#) on page 284

[LIN serial bus configuration menu](#) on page 288

[MIL-STD-1553 serial bus menu](#) on page 290

[Parallel Bus configuration menu](#) on page 292

[RS232 serial bus menu](#) on page 295

[SENT serial bus configuration menu](#) on page 297

[SPI serial bus configuration menu](#) on page 299

[SPMI serial bus configuration menu](#) on page 301

[USB serial bus configuration menu](#) on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

**LIN serial bus  
configuration menu**

Use this menu (optional) to set up and display a LIN (Local Interconnect Network) serial bus waveform.

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.**

<b>Field or control</b>	<b>Description</b>
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to LIN.
<b>Source</b>	Set the signal source from available analog or digital channels.
<b>Threshold</b>	Set the threshold level to define a logic high level.
<b>Polarity</b>	Select the polarity to match the LIN bus being acquired.
<b>LIN Standard</b>	Select the standard to match the LIN bus being acquired.
<b>Bit rate</b>	Sets the bit rate.  To enter a custom bit rate, tap Custom and enter the custom bit rate in the Custom Rate input box.
<b>Include Parity Bits with ID</b>	Set to Yes to include parity bits with the ID.
<b>Sample Point</b>	Sets the sample point from 5% to 95% of the position within the bit period or the unit interval.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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 274

*Audio serial bus configuration menu* on page 275

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*FlexRay serial bus configuration menu* on page 282

*I2C serial bus configuration menu* on page 284

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

### **MIL-STD-1553 serial bus menu**

Use the MIL-STD-1553 bus menu (optional) to set up and decode a MIL-STD-1553 aeronautic network serial data bus waveform.

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
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>MIL-STD-1553</b> .
<b>Polarity</b>	Select the polarity to match the MIL-STD-1553 bus being acquired.
<b>Source</b>	Select the MIL-STD-1553 signal source.
<b>High Threshold, Low Threshold</b>	Sets the valid high and low threshold values for the signal source.
<b>RT Maximum</b>	Sets the maximum valid response time (RT) for a command.
<b>RT Minimum</b>	Sets the minimum valid response time (RT) for a command.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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 274

*Audio serial bus configuration menu* on page 275

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

## 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
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to Parallel to define a parallel bus.
<b>Clocked Data</b>	Toggles Yes or No to use a clock signal to recover the data bits from the bus inputs.
<b>Clock Source</b>	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.
<b>Polarity</b>	Sets the clock signal edge to use for timing reference.  Only available when Clocked Data is set to Yes.
<b>Threshold</b>	Sets the threshold value to determine high logic value.  Only available when Clocked Data is set to Yes.
<b>Define Inputs</b>	Opens a Parallel Bus - Define Inputs configuration menu to set the signal sources and the bit order (MSB to LSB) for the bus. See <a href="#">Parallel Bus - Define Inputs menu</a> on page 294.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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 274

*Audio serial bus configuration menu* on page 275

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*FlexRay serial bus configuration menu* on page 282

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

**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
<b>Parallel bus definition list</b>	<p>Lists the signal source and thresholds of selected channels or waveforms. The MSB is at the top of the list.</p> <p>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.</p> <p>Use the arrow buttons to the right of the field to move a selected signal up or down in the list.</p> <p>To remove a signal from the Parallel bus (and return it to the Sources list), tap on the signal source button.</p> <p>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.</p>
<b>Sources</b>	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.
<b>Set All Thresholds</b>	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 (optional) to set up and display an RS232 serial bus waveform.

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
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>RS232</b> .
<b>Bit Rate</b>	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.
<b>Source</b>	Set the signal source from available analog or digital channels.
<b>Threshold</b>	Set the threshold level to define a logic high level.
<b>Polarity</b>	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.
<b>Data Bits</b>	Set the number of bits that define a data packet for your RS232 bus.
<b>Parity</b>	Set the parity to match the RS232 bus being acquired.
<b>Packet View</b>	Set to On to show decoded packet level information on the bus waveform.
<b>End of packet</b>	select the appropriate end of packet value to match the RS232 bus being acquired.  Available when Packet View = On.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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 274

*Audio serial bus configuration menu* on page 275

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*FlexRay serial bus configuration menu* on page 282

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

**SENT serial bus configuration menu**

Use the SENT bus menu (optional) to set up and display a SENT (Single Edge Nibble Transmission) serial bus waveform. Requires option SRAUTOSEN.

To add a SENT serial bus waveform to the display:

1. Tap **Add New Bus** on the Settings bar.
2. Double-tap the new bus badge to open the bus configuration menu.
3. Set the **Bus Type** to **SENT**.
4. Use the menu fields and controls to configure the **SENT** bus parameters.

To change the settings on an existing CAN serial bus waveform, double-tap the **SENT** bus badge to open the configuration menu.

**SENT serial bus menu fields and controls.**

Field or control	Description
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	Enter a label for the bus. The default label is the selected bus type.  To enter label text, double-tap the field and use the virtual keyboard to enter label text, or tap the field and enter text from an attached keyboard.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>SENT</b> to set up and display a SENT serial bus waveform.
<b>Source</b>	Select the signal source from the listed analog and digital channels.
<b>Threshold</b>	Sets the high/low logic transition level.
<b>Polarity</b>	Sets the signal edge to use to determine the width of the clock tick pulse.  <b>Normal</b> (default) corresponds to a falling edge polarity that determines how many ticks wide a pulse is.  <b>Inverted</b> corresponds to a rising edge polarity.
<b>Clock Tick</b>	Sets the time period of the clock tick. The valid range is from 1 $\mu$ s to 300 $\mu$ s. The default is 3 $\mu$ s.
<b>Tick Tolerance</b>	Sets the tolerance, as a percent, that is acceptable for the clock tick signal to be recognized. The valid tolerance range is from 1% to 30%. The default tolerance is 20%.
<b>Fast Data Channels</b>	Sets the number of fast data channels. The default is 2.
<b>Data Nibbles</b>	Sets the number of data nibbles to detect in the serial signal ( <b>3</b> , <b>4</b> , or <b>6</b> ).  Available when <b>Fast Data Channels = 1</b> .
<b>Channel Widths (C1/C2)</b>	Sets the number of bits per channel when using two fast data channels ( <b>12/12</b> , <b>14/10</b> , or <b>16/8</b> ).  Available when <b>Fast Data Channels = 2</b> .
<b>Pause Pulse</b>	Sets the instrument to detect a Pause pulse in the serial data. The default value is Yes.
<b>Slow Channel</b>	Sets the slow channel characteristics. Tap and select from the available list of slow channel types. The default value is None.
<b>Display</b>	<b>Bus</b> sets the waveform view to show just the decoded bus information.  <b>Bus and Waveform</b> 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
<b>Decode Format</b>	Sets the decode format used to display the bus information. Formats are <b>Hex</b> , <b>Binary</b> , <b>Mixed Decimal</b> , and <b>Mixed Hex</b> .

**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 274

[Audio serial bus configuration menu](#) on page 275

[Ethernet serial bus menu](#) on page 280

[FlexRay serial bus configuration menu](#) on page 282

[I2C serial bus configuration menu](#) on page 284

[LIN serial bus configuration menu](#) on page 288

[MIL-STD-1553 serial bus menu](#) on page 290

[Parallel Bus configuration menu](#) on page 292

[RS232 serial bus menu](#) on page 295

[SPI serial bus configuration menu](#) on page 299

[SPMI serial bus configuration menu](#) on page 301

[USB serial bus configuration menu](#) on page 303

**See also.**

[Bus Trigger configuration](#) on page 488

[Bus Search configuration menus](#) on page 320

**SPI serial bus configuration menu**

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

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
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>SPI</b> .
<b>Framing</b>	Set to Slave Select (SS) or Idle framing modes.
<b>SCLK Input</b>	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.
<b>SS Input</b>	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.
<b>SDA Input</b>	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.
<b>Word Size</b>	Enter the word size, in bits. Common word sizes are 8, 16, and 32.
<b>Bit Order</b>	Set to most significant bit first (MS) or least significant bit first (LS).
<b>Idle Time (Framing = )</b>	Set the idle frame time.  Available when Framing = Idle.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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<sup>2</sup>C, 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 274

*Audio serial bus configuration menu* on page 275

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*FlexRay serial bus configuration menu* on page 282

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPMI serial bus configuration menu* on page 301

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

**SPMI serial bus configuration menu**

Use the SPMI bus menu (optional) to set up and display an SPMI (System Power Management Interface) serial bus waveform. Requires option SRPM.

To add an SPMI serial bus waveform to the display:

1. Tap **Add New Bus** on the Settings bar.
2. Set the **Bus Type** to **SPMI**.
3. Use the menu fields and controls to configure the **SPMI** bus parameters.

To change the settings on an existing SPMI serial bus waveform, double-tap the **SPMI** bus badge to open the configuration menu.

**SPMI serial bus menu fields and controls.**

Field or control	Description
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>SPMI</b> to set up and display a SPMI bus waveform.
<b>SCLK Input</b>	Sets the signal <b>Source</b> and the high/low logic transition <b>Threshold</b> level for the <b>SCLK</b> input signal.
<b>SDATA Input</b>	Sets the signal <b>Source</b> and the high/low logic transition <b>Threshold</b> level for the <b>SDATA</b> input signal.
<b>Display</b>	<b>Bus</b> sets the waveform view to show just the decoded bus information.  <b>Bus and Waveforms</b> 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.
<b>Decode Format</b>	Sets the decode format used to display the bus information. Formats are <b>Hex</b> , <b>Binary</b> , and <b>Mixed Hex</b> .



**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 capabilities to the **Trigger** and **Search** menus.

*ARINC 429 serial bus menu* on page 274

*Audio serial bus configuration menu* on page 275

*Ethernet serial bus menu* on page 280

*FlexRay serial bus configuration menu* on page 282

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*USB serial bus configuration menu* on page 303

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

**USB serial bus configuration menu**

Use the USB bus menu (optional) to set up and display an USB 2.0 (Universal Serial Bus) waveform.

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
<b>Display</b>	Turns on or off displaying the bus on the Waveform view.
<b>Label</b>	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.
<b>Position</b>	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.
<b>Set to 0</b>	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
<b>Bus Type</b>	Set to <b>USB</b> .
<b>Speed</b>	Set the speed to match the USB bus you are acquiring.  Selecting High Speed (480 Mbps) sets the Signal Type to Differential.
<b>Signal type</b>	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.
<b>Source</b>	Select the channel source for the signal from a differential probe.  Only available when Signal Type = Diff.
<b>High Threshold</b>	Set the high threshold level for the differential signal.
<b>Low Threshold</b>	Set the low threshold level for the differential signal.
<b>D+ Input</b>	Select the channel source and set the threshold level for the Data+ signal.  Only available when Signal Type = Single Ended.
<b>D- Input</b>	Select the channel source and set the threshold level for the Data- signal.  Only available when Signal Type = Single Ended.
<b>Display Format</b>	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
<b>Decode Format</b>	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 274

*Audio serial bus configuration menu* on page 275

*CAN serial bus configuration menu* on page 277

*Ethernet serial bus menu* on page 280

*FlexRay serial bus configuration menu* on page 282

*I2C serial bus configuration menu* on page 284

*LIN serial bus configuration menu* on page 288

*MIL-STD-1553 serial bus menu* on page 290

*Parallel Bus configuration menu* on page 292

*RS232 serial bus menu* on page 295

*SENT serial bus configuration menu* on page 297

*SPI serial bus configuration menu* on page 299

*SPMI serial bus configuration menu* on page 301

**See also.**

*Bus Trigger configuration* on page 488

*Bus Search configuration menus* on page 320

## 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
<b>Sources</b>	Sets the input sources for the plot. The number of source fields listed depends on the plot.
<b>XY, XYZ, Eye Diagram</b>	Select the plot type to add to the screen. <b>Eye Diagram</b> adds a TIE measurement badge to the Results bar and adds the eye diagram to the screen.
<b>Add</b>	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 444.

### Closing a plot view

To close (delete) a Plot view, tap the **X** in the upper right corner of the view. Deleting the Measurement badge that enabled the plot also closes the plot.

## Add Results Table configuration menu

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

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

To add a result table to the screen:

1. Tap **Add New... Results Table**.
2. Select a table type.
3. Tap **Add**. You can also double-tap a table type to add it to the screen.

### Add Results Table menu fields and controls

Field or control	Description
<b>Measurements</b>	Display a table of all the measurements in the Results bar.
<b>Search</b>	Display a table of all defined searches with each search shown on its own tab.
<b>Bus Decode</b>	Display a table of the bus decode results.
<b>Harmonics/Custom</b> (option 5-PWR, 6-PWR only)	Displays a table of the Harmonics measurement results. Custom displays a table of the Frequency Response Analysis measurement results.  Custom is available only for Inrush Current, Input Capacitance, Turn-on Time, Turn-off time, Control loop response, and PSRR power measurements.
<b>Add</b>	Displays the selected table type. You can add as many tables as you want to the screen.

## Results Tables operations overview

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

### Results Table - general operations.

- Double-tap anywhere on a results table to open its configuration menu.
- To scroll up and down in a results table, tap the table and 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 308

[Bus Decode Results table configuration menu](#) on page 313

[Search Results table menu](#) on page 311

[Harmonics/Custom Results table configuration menu \(option 5-PWR, 6-PWR only\)](#) on page 315

## 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
<b>Column Visibility panel</b>	
<b>Label</b>	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.
<b>Peak-to-Peak</b>	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.
<b>Column Resolution panel</b>	
<b>Measurement column resolution</b>	Sets the resolution (number of digits to display) in each column.
<b>Set to Defaults</b>	Sets all columns to show 5 digit readout resolution.
<b>Other panel</b>	
<b>Statistics</b>	<p>Sets the amount of statistical information you want to display for each measurement.</p> <p><b>Both</b> shows both the accumulated results and the current acquisition results.</p> <p><b>All Acquisitions</b> shows statistics for all acquisitions from the last action that cleared acquisition memory.</p> <p><b>Current Acquisition</b> shows the statistics for just the current acquisition.</p>
<b>Cycle-to-Cycle Variation</b>	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.
<b>Row Height Optimized For</b>	<p>Select Touch to increase table row height by ~75% for easier touch manipulation. Uses more space on the screen.</p> <p>Select Mouse to decrease row height. Uses less space on the screen. Mouse is the default setting.</p>
<b>Save Table</b>	Opens a menu with which to save the results table data to a file. See <a href="#">Save As configuration menu (Measurement Results Table)</a> on page 310.

**Measurements Results Table operations.**

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

**Save As configuration menu (Measurement Results Table)**

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

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

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



**Save As menu, Measurement Results table fields and controls.**

Field or control	Description
<b>Save Location</b>	<p>Sets the location to which to save the file. The default value is the last location to which a file was saved.</p> <p>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.</p> <p>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.</p>
<b>Browse</b>	<p>Tap to open the Browse Save As Location dialog, to navigate to and select the location to which to save the file. See <a href="#">Browse Save As Location configuration menu</a> on page 414.</p>
<b>File Name</b>	<p>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).</p> <p>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.</p>
<b>Save as Type</b>	<p>You can only save table results as comma separated values (.csv) files.</p>

**Search Results table menu**

Use this menu to configure the content of the Search results table.

To open the Search Results Table configuration menu, double-tap anywhere in the Search Results table. If there are multiple search results tables, tap the tab of the search table to configure and then double-tap anywhere in that table.

**Search Results Table menu.**

Field or control	Description
<b>Column Visibility panel</b>	
Check boxes to select columns to display	Select or clear individual check boxes to add or remove that column from the table.
<b>Other panel</b>	
<b>Location Timestamp Resolution</b>	<p>Sets the search mark timestamp resolution to display. The timestamp shows the time of each mark relative to the first acquired search mark.</p> <p><b>Short</b> shows a rounded version of the timestamp data.</p> <hr/> <p><b>NOTE.</b> <i>The actual timestamp data is not truncated by the Short setting, and is retained in the acquisition data.</i></p> <hr/> <p><b>Precision</b> adds additional columns to the table to display the full timestamp value by individual time units (seconds, milliseconds, microseconds, and so on).</p>
<b>Delta resolution</b>	<p>Sets the timestamp resolution to display for mark deltas. Delta shows the time difference between each search mark.</p> <p><b>Short</b> shows the delta time between marks as a rounded version of the timestamp data.</p> <hr/> <p><b>NOTE.</b> <i>The actual timestamp data is not truncated by the Short setting, and is retained in the acquisition data.</i></p> <hr/> <p><b>Precision</b> adds additional columns to the table to display the full delta timestamp value by individual time units (seconds, milliseconds, microseconds, and so on).</p>
<b>Save Table</b>	Opens a menu with which to save the results table data to a file. See <a href="#">Save As configuration menu (Measurement Results Table)</a> on page 310.

**Search Results Table operations.**

- If there are multiple searches, each search has a tab in the table. Tap the tab of the search to display and/or configure, and then double-tap anywhere in that table to open the configuration menu for that table.
- Double-tap anywhere on a results table to open its configuration menu.
- If you add or delete a search to the Results bar, that search is automatically added to or deleted from an existing Search Results table.
- 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. If the column width change forces the truncation of data, hovering over a cell displays the full information content of that cell (subject to significant digits limitations on numeric entries).
- You cannot sort the contents of a column.
- You cannot change the order of rows.

**Bus Decode Results table configuration menu**

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

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

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

**Bus Decode Results table menu.**

Field or control	Description
<b>Column Visibility panel</b>	
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.
<b>Other panel</b>	
<b>Save Table</b>	Opens the Save As dialog to let you save the table data to a comma-separated values (.csv) file. See <a href="#">Save As configuration menu (Bus Decode Results table)</a> on page 314.

### Bus Decode Results Table operations.

- Each bus in a Bus Decode Results table has its own tab. Tap a tab to show the results for that bus.
- Selecting a Bus in the Bus Decode Table configuration menu does not select and display the tab for that bus. Select a tab before configuring the table for that tab.
- Double-tap anywhere on a results table to open the configuration menu for that tab.
- 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.
- 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.
- You cannot sort the contents of a column.

### Save As configuration menu (Bus Decode Results table)

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

To open the Save As configuration menu:

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

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

Field or control	Description
<b>Save Location</b>	<p>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.</p> <p>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.</p> <p>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.</p>
<b>Browse</b>	Tap to open the Browse Save As Location configuration menu, to navigate to and select the location to which to save the file. See <a href="#">Browse Save As Location configuration menu</a> on page 414.
<b>File Name</b>	<p>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.</p> <p>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.</p>
<b>Save as Type</b>	You can only save table results as comma separated values (.csv) files.

**Harmonics/Custom  
Results table  
configuration menu  
(option 5-PWR, 6-PWR  
only)**

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

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

**NOTE.**

- *If multiple harmonics results tables are displayed, each result is in a separate tab. Tap the harmonics tab of interest to open that table, and then double-tap in the Harmonics results table area to open that table's configuration menu.*
- *Custom is displayed only for Control Loop Response and Power Supply Rejection Ratio measurements.*

**Harmonics/Custom Table configuration menu.**

Field or control	Description
<b>Table Settings panel</b>	
<b>Column Visibility</b>	Select or clear individual check boxes to add or remove that column from the table.
<b>Units panel:</b> Available only for Harmonics measurement.	
<b>Units</b>	Sets the vertical scale used to display measurement data. Select Log or Linear.
<b>Harmonics</b>	Select <b>All</b> to show all harmonics, <b>Odd</b> to show just odd harmonics, or <b>Even</b> to show just even harmonics in the table.  <i><b>NOTE.</b> Changing this field also changes the harmonics bars shown in the Harmonics bar plot.</i>
<b>Save panel</b>	
<b>Save Table</b>	Opens a menu with which to save the Harmonics/Custom Results table data to a comma-separated value format file. See <a href="#">Save As configuration menu (Harmonics/Custom Results Table)</a> on page 316.

**Save As configuration menu (Harmonics/Custom Results Table)**

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

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

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

**Save As configuration menu (Harmonics/Custom Results Table).**

Field or control	Description
<b>Save Location</b>	<p>Sets the location to save the file. The default value is the last location to which a file was saved.</p> <p>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.</p> <p>You can also use the Browse button to open a menu to navigate to and select the location to which to save the file.</p> <p>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.</p>
<b>Browse</b>	<p>Tap to open the Browse Save As Location dialog, to navigate to and select the location to which to save the file. See <a href="#">Browse Save As Location configuration menu</a> on page 414.</p>
<b>File Name</b>	<p>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).</p> <p>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.</p>
<b>Save as Type</b>	<p>You can only save a Harmonics/Custom Results table to a comma-separated value (csv) format file.</p>

**Annotation and navigation on waveform plots/data and results table (optional)**

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,  $R_{DS(on)}$ , 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** or **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** or **Max** buttons to navigate to the measurement region containing the minimum and maximum measurement values.

---

**NOTE.**

- $R_{DS(on)}$  supports an annotation feature that marks the minimum value of  $R_{DS(on)}$  with a line in each switching cycle.
  - When  $R_{DS(on)}$  is added, a Math waveform is created. Math (resistance curve) equation can be a  $V/I$  or  $\Delta V/\Delta I$  based on the measurement configuration, where  $V/I$  is static and  $\Delta V/\Delta I$  is dynamic in nature.
  - You can use the Measurement badge **Previous** and **Next** buttons to navigate from switching cycle to switching cycle. You can also view the minimum  $R_{DS(on)}$  value marked in the respective switching cycle.
  - By default,  $R_{DS(on)}$  places the annotation lines on the switching cycle where the minimum  $R_{DS(on)}$  value occurs. When you select the measurement results badge, the navigation function enables zoom mode and centers the minimum  $R_{DS(on)}$  value for the switching cycle in the display.
-



### Navigation on Bar Graph and Harmonics Results Table (optional)

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 320

[Edge Search configuration menu](#) on page 355

[Logic search configuration menu](#) on page 356

[Pulse Width Search configuration menu](#) on page 359

[Rise/Fall Time Search configuration menu](#) on page 361

[Runt Search configuration menu](#) on page 363

[Setup and Hold Search configuration menu](#) on page 365

[Timeout Search configuration menu](#) on page 368

[Window Search configuration menu](#) on page 369

[DDR Read Search configuration menu \(Requires option SUP6-DBDDR3\)](#) on page 372

[DDR Write Search configuration menu \(Requires option SUP6-DBDDR3\)](#) on page 374

[DDR Read & Write Search configuration menu \(Requires option SUP6-DBDDR3\)](#) on page 376

## 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.

To create a new Bus search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Bus**.
3. Select the bus **Source**.
4. Use the search menu fields to set the search parameters.

To change the settings of an existing search, double-tap the search badge to open its configuration menu and make necessary changes.

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

[ARINC 429 serial bus search configuration menu](#) on page 321

[Audio serial bus search configuration menu](#) on page 323

[CAN serial bus search configuration menu \(when not using a .dbc symbol definition file\)](#) on page 324

[Ethernet serial bus search configuration menu](#) on page 328

[FlexRay serial bus search configuration menu](#) on page 331

[I2C serial bus search configuration menu](#) on page 333

[I3C serial bus Search configuration menu](#) on page 334

[LIN serial bus search configuration menu](#) on page 344

[MIL-STD-1553 serial bus search configuration menu](#) on page 345

[Parallel bus search configuration menu](#) on page 347

[RS-232 serial bus search configuration menu](#) on page 347

[SENT serial bus search configuration menu](#) on page 348

[SPI serial bus search configuration menu](#) on page 351

[SPMI serial bus search configuration menu](#) on page 351

[USB serial bus search configuration menu](#) on page 353

**Other search types.** *Edge Search configuration menu* on page 355

*Logic search configuration menu* on page 356

*Pulse Width Search configuration menu* on page 359

*Rise/Fall Time Search configuration menu* on page 361

*Runt Search configuration menu* on page 363

*Setup and Hold Search configuration menu* on page 365

*Timeout Search configuration menu* on page 368

*Window Search configuration menu* on page 369

*DDR Read Search configuration menu (Requires option SUP6-DBDDR3)* on page 372

*DDR Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 374

*DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 376

**ARINC 429 serial bus search configuration menu**

Use the ARINC 429 Search configuration menu to define conditions to search for and mark on an ARINC 429 bus waveform. You can have multiple searches on the same bus.

---

**NOTE.** *Requires option SRAERO.*

---

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the <b>ARINC 429</b> bus to search.
<b>Mark When</b>	Sets the type of information for which to search.
<b>Trigger When Label</b>	Sets the label condition for which to search.
<b>Label</b>	<p>Sets the label pattern for which to search.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark When = Label</b> or <b>Label &amp; Data</b>.</p>

Field or control	Description
<b>Label Low</b>	<p>Sets the low value of the label pattern range for which to search.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark When = Label</b> and <b>Mark When Label = Inside Range</b> or <b>Outside Range</b>.</p>
<b>Label High</b>	<p>Sets the high value of the label pattern range for which to search.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark When = Label</b> and <b>Mark When Label = Inside Range</b> or <b>Outside Range</b>.</p>
<b>Trigger When Data</b>	<p>Sets the data condition for which to search.</p>
<b>Data</b>	<p>Sets the data pattern for which to search.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark When = Data</b> or <b>Label &amp; Data</b>.</p>
<b>Data Low</b>	<p>Sets the low value of the data pattern range for which to search.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark When = Data</b> and <b>Mark When Data = Inside Range</b> or <b>Outside Range</b>.</p>
<b>Data High</b>	<p>Sets the high value of the data pattern range for which to search.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark When = Data</b> and <b>Mark When Data = Inside Range</b> or <b>Outside Range</b>.</p>
<b>SSM</b>	<p>Sets the Sign/Status Matrix (SSM) bit condition for which to search.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark When = Data</b> or <b>Label &amp; Data</b>.</p>

Field or control	Description
<b>SDI</b>	<p>Sets the Source/Destination Identifier (SDI) bit condition for which to search.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark When = Data</b> or <b>Label &amp; Data</b>.</p>
<b>Error Type</b>	<p>Sets the error condition for which to search.</p> <p>Available when <b>Mark When = Error</b>.</p>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

### Audio serial bus search configuration menu

Use the Audio Search configuration menu to define conditions to search for and mark on an Audio bus waveform. You can have multiple searches on the same bus.

**NOTE.** Requires option *SRAUDIO*.

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the <b>Audio</b> bus to search.
<b>Mark On</b>	Select the type of information for which to search.
<b>Data</b>	<p>Sets the data pattern for which to search. Use in conjunction with the Mark When field to specify the exact search condition.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Data</b>.</p>
<b>Word</b>	<p>Sets the audio word channel for which to search.</p> <p>Only available when <b>Mark On = Data</b>.</p>

Field or control	Description
<b>Mark When</b>	<p>Sets the mark when condition for the specified data pattern.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a high and low boundary pattern for the specified search range.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Data</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**CAN serial bus search configuration menu (when not using a .dbc symbol definition file)**

Use the CAN Search configuration menu to define conditions to search for and mark on an CAN bus waveform. This menu supports searches for CAN bus events when a .dbc symbol definition file is not loaded.

---

**NOTE.** Requires option *SRAUTO*.

---

**NOTE.** See [CAN serial bus search configuration menu \(when using a .dbc symbol definition file\)](#) on page 326 for CAN serial bus searches that use a loaded .dbc symbol definition file.

---

Field or control	Description
<b>Display</b>	Enables or disables displaying waveform search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the <b>CAN</b> bus that you want to search.

Field or control	Description
<b>Mark On</b>	<p>Sets the type of information for which to search and mark on the bus waveform acquisition.</p> <p><b>Start of Frame</b> puts a search mark at each occurrence of a CAN frame start.</p> <p><b>Type of Frame</b> puts a search mark at each occurrence of the frame type as set in the Frame Type field.</p> <p><b>Identifier</b> puts a search mark at each occurrence of a CAN identifier frame as set in the <b>Direction</b>, <b>Identifier Format</b>, and <b>Identifier</b> fields. Not available when the bus badge <b>Decode Format</b> is set to <b>Symbolic</b>.</p> <p><b>Data</b> puts a search mark at the occurrence of the specified data fields (<b>Data Bytes</b>, <b>Data Offset</b>, <b>Mark When</b>, and <b>Data</b>). Not available when the bus badge <b>Decode Format</b> is set to <b>Symbolic</b>.</p> <p><b>Id &amp; Data</b> puts a search mark at each occurrence of the specified Id and data fields. Not available when the bus badge <b>Decode Format</b> is set to <b>Symbolic</b>.</p> <p><b>End of Frame</b> puts a search mark at each occurrence of a CAN frame end.</p> <p><b>Error</b> puts a search mark at each occurrence of the specified error condition (<b>Missing Ack</b>, <b>Bit Stuffing</b>, <b>FD Form Error</b>, or <b>Any Error</b>).</p>
<b>Frame Type</b>	<p>Sets the frame type for which to search and mark on the bus waveform (<b>Data Frame</b>, <b>Remote Frame</b>, <b>Error Frame</b>, <b>Overload Frame</b>).</p> <p>Only available when <b>Mark On</b> = <b>Type of Frame</b>.</p>
<b>Error Type</b>	<p>Sets the error type for which to search and mark on the bus waveform. (<b>Missing Ack</b>, <b>Bit Stuffing</b>, or <b>Any Error</b>).</p> <p>Only available when <b>Mark On</b> = <b>Error</b>.</p>
<b>Direction</b>	<p>Sets the Identifier's transfer direction (<b>Read</b>, <b>Write</b>, or <b>Either</b>) for which to search.</p> <p>Only available when <b>Mark On</b> = <b>Identifier</b>.</p>
<b>Identifier Format</b>	<p>Sets the CAN identifier format bit length for which to search (<b>Standard</b> 11-bit, or <b>Extended</b> 29-bit for CAN 2.0B).</p> <p>Only available when <b>Mark On</b> = <b>Identifier</b> or <b>ID &amp; Data</b>.</p>
<b>Identifier</b>	<p>Sets the identifier pattern for which to search. The number of bits shown depends on the <b>Identifier Format</b> setting.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On</b> = <b>Identifier</b> or <b>ID &amp; Data</b>.</p>

Field or control	Description
<b>Data Bytes</b>	Sets the number of data bytes for which to search (one to eight bytes). Use the <b>A</b> knob to change the value.  Only available when <b>Mark On = Data</b> or <b>ID &amp; Data</b> .
<b>Data Offset</b>	Sets the data offset value. Use the <b>A</b> knob to change the value.  Only available when <b>Mark On = Data</b> or <b>ID &amp; Data</b> .
<b>Mark When</b>	Sets the mark when condition (mark the waveform when the waveform is =, ≠, <, >, ≤, or ≥ the data conditions).  Only available when <b>Mark On = Data</b> or <b>ID &amp; Data</b> .
<b>Data</b>	Sets the data pattern for which to search. The number of bits shown depends on the <b>Data Bytes</b> setting.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Data</b> or <b>ID &amp; Data</b> .
<b>A, B knob controls</b>	Use the <b>A</b> knob to select (highlight) the digit to change.  Use the <b>B</b> knob to change the value of the digit.
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**CAN serial bus search configuration menu (when using a .dbc symbol definition file)**

Use the CAN Search configuration menu to define conditions to search for and mark on an CAN bus waveform. This menu supports searches for CAN bus events based on labels derived from a loaded .dbc symbol definition file.

---

**NOTE.** Requires option 5/6-SRAUTO.

---

Field or control	Description
<b>Display</b>	Enables or disables displaying waveform search marks for this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the <b>CAN</b> bus that you want to search and mark.



Field or control	Description
<b>Mark On</b>	<p>Sets the type of information for which to search and mark on the bus waveform acquisition.</p> <p><b>Start of Frame</b> puts a search mark at each occurrence of a CAN frame start.</p> <p><b>FD Bits</b> puts a search mark at each occurrence of the specified <b>BRS Bit</b> (Bit Rate Switch) and <b>ESI Bit</b> (Error State Indicator) states. Not available when the bus badge <b>Decode Format</b> is set to <b>Symbolic</b>.</p> <p><b>End of Frame</b> puts a search mark at each occurrence of a CAN frame end.</p> <p><b>Error</b> puts a search mark at each occurrence of the specified error condition (<b>Missing Ack</b>, <b>Bit Stuffing</b>, <b>FD Form Error</b>, or <b>Any Error</b>).</p> <p><b>Symbolic</b> puts a search mark at each occurrence of a specified message or signal condition as defined by an installed .dbc symbol definition file. Only available when the bus badge <b>Decode Format</b> is set to <b>Symbolic</b>.</p>
<b>Configure</b>	<p>Opens the <b>CAN DBC Symbolic Configuration</b> menu, which sets the CAN bus search parameters based on the message and signal labeling derived from an installed .dbc symbol definition file. See <a href="#">CAN serial bus search configuration menu (when using a .dbc symbol definition file)</a> on page 326.</p>
<b>Message, Signal, and Value fields</b>	<p>These fields display the actual search data values for the <b>Message</b>, <b>Signal</b>, and <b>Value</b> fields as set in the <b>CAN DBC Symbolic Configuration</b> menu. These fields are shown for reference and are not editable.</p>
<b>BRS Bits</b>	<p>Sets the <b>BRS</b> (Bit Rate Switch) bit state for which to search and mark at each occurrence on the bus waveform.</p> <p>Only available when <b>Mark On</b> = <b>FD Bits</b>.</p>
<b>ESI Bits</b>	<p>Sets the <b>ESI</b> (Error State Indicator) bit state for which to search and mark at each occurrence on the bus waveform.</p> <p>Only available when <b>Mark On</b> = <b>FD Bits</b>.</p>
<b>Error Type</b>	<p>Sets the error type for which to search and mark on the bus waveform. (<b>Missing Ack</b>, <b>Bit Stuffing</b>, or <b>Any Error</b>).</p> <p>Only available when <b>Mark On</b> = <b>Error</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit to change.</p> <p>Use the <b>B</b> knob to change the value of the digit.</p>
<b>Copy Trigger Settings to Search</b>	<p>Sets the search criteria to match the current oscilloscope trigger settings.</p>
<b>Copy Search Settings to Trigger</b>	<p>Sets the current oscilloscope trigger settings to match the search criteria.</p>

**CAN DBC Symbolic Configuration menu**

Use this menu to set the CAN .dbc symbol definitions for which to search. This menu is accessed from a CAN Bus Search menu when a .dbc symbol definition file is loaded on the instrument. Load the .dbc file when you first add and configure the CAN bus.

---

**NOTE.** Requires option 5/6-SRAUTO.

---

Field or control	Description
<b>Symbolic Type</b>	Sets whether to search for <b>Message</b> CAN or search on both <b>Message and Signal</b> code.
<b>Message</b>	Sets the CAN message label for which to search. Use the drop-down list to select from message labels derived from the installed .dbc file.
<b>Signal</b>	Sets the CAN signal label for which to search. Use the drop-down list to select from signal labels derived from the installed .dbc file.
<b>Signal Encoded</b>	Enables searching for encoded signal values (when selected) or specific values (when not selected). Use the drop-down list to select from encoded signal types derived from the installed .dbc file.  Value searching is always available for a signal. Signals, however, are not necessarily encoded, so searches on signal encoding are not always available.
<b>Value</b>	Sets a specific <b>Signal</b> value for which to search. Use knob A to set a value, or double-tap the field and enter a value using the virtual keyboard.

**See also.** [CAN serial bus search configuration menu \(when using a .dbc symbol definition file\)](#) on page 326

[CAN serial bus configuration menu](#) on page 277

**Ethernet serial bus search configuration menu**

Use the Ethernet Search configuration menu to define conditions to search for and mark on an Ethernet bus waveform. You can have multiple searches on the same bus.

---

**NOTE.** Requires option SRENET.

---

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the Ethernet bus that you want to search.
<b>Mark On</b>	Select the type of information for which to search.

Field or control	Description
<b>MAC Address Destination, MAC Address Source</b>	<p>Sets the MAC destination and/or source address pattern for which to search.</p> <p>Tap a <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Mac Addresses</b>.</p>
<b>Q-Tag</b>	<p>Sets the Q-tag pattern for which to search.</p> <p>Tap a <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Q-Tag Control Information</b>.</p>
<b>MAC Length/Type</b>	<p>Enter the MAC length or type pattern for which to search. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = MAC Length/Type</b>.</p>
<b>IP Protocol</b>	<p>Sets the IP protocol pattern for which to search.</p> <p>Tap a <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = IP Header</b>.</p>
<b>Source, Destination Address</b>	<p>Sets the source and/or destination IP address pattern for which to search.</p> <p>Tap a <b>Hex</b> or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = IP Header</b>.</p>
<b>Source Port, Destination Port</b>	<p>Sets the source and/or destination TCP header port pattern for which to search.</p> <p>Tap a <b>Binary</b>, <b>Hex</b> or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = TCP Header</b>.</p>

Field or control	Description
<b>Sequence Number</b>	<p>Sets the TCP header sequence number pattern for which to search.</p> <p>Tap a <b>Binary</b>, <b>Hex</b> or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = TCP Header</b>.</p>
<b>Ack Number</b>	<p>Sets the TCP header ack number pattern for which to search.</p> <p>Tap a <b>Binary</b>, <b>Hex</b> or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = TCP Header</b>.</p>
<b>Data Bytes</b>	<p>Sets the number of client data bytes for which to search (one to sixteen bytes). Use the <b>A</b> knob to change the value.</p> <p>Only available when <b>Mark On = Client Data</b>.</p>
<b>Byte Offset</b>	<p>Sets the client data byte offset (<b>Don't Care</b> or the number of bytes). Tap the field and use the <b>A</b> knob to change the value.</p> <p>Only available when <b>Mark On = Client Data</b>.</p>
<b>Client Data</b>	<p>Sets the data pattern for which to search. The number of bits shown depends on the <b>Data Bytes</b> setting. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition.</p> <p>Tap the <b>Binary</b>, <b>Hex</b> or <b>ASCII</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Client Data</b>.</p>
<b>Mark When</b>	<p>Sets the mark when condition.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a high and low boundary pattern for the specified search range.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = MAC Length/Type</b> or <b>Client Data</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**FlexRay serial bus search configuration menu**

Use the FlexRay Search configuration menu to define conditions to search for and mark on an FlexRay bus waveform. You can have multiple searches on the same bus.

**NOTE.** Requires option *SRAUTO*.

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the FlexRay bus for which to search.
<b>Mark On</b>	Select the type of information for which to search.
<b>Indicator Bits</b>	Select the defined indicator bits type for which to search from the drop-down list.  Only available when <b>Mark On = Indicator Bits</b> .
<b>Indicator Bits</b>	Enter the indicator bits for which to search.  Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Header Fields</b> .
<b>Identifier</b>	Enter the frame identifier pattern for which to search.  Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Frame ID</b> or <b>Header Fields</b> .
<b>Cycle Count</b>	Enter the cycle count pattern for which to search. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition.  Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Cycle Count</b> or <b>Header Fields</b> .
<b>Payload Length</b>	Enter the payload length pattern for which to search. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition.  Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Header Fields</b> .

Field or control	Description
<b>Header CRC</b>	<p>Enter the header CRC pattern for which to search. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Header Fields</b>.</p>
<b>Data Bytes</b>	<p>Enter the number of data bytes for which to search (one to sixteen bytes). Use the <b>A</b> knob to change the value.</p> <p>Only available when <b>Mark On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>Data</b>	<p>Enter the data pattern for which to search. The number of bits shown depends on the <b>Data Bytes</b> setting. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>Byte Offset</b>	<p>Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the <b>A</b> knob to change the value.</p> <p>Only available when <b>Mark On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>Mark When</b>	<p>Sets the mark when condition.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a high and low boundary pattern for the specified search range.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Frame ID, Cycle Count, Data, or Identifier &amp; Data</b>.</p>
<b>Frame Type</b>	<p>Sets the end of frame type for which to search.</p> <p>Only available when <b>Mark On = End of Frame</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**I2C serial bus search configuration menu**

Use the I2C Search configuration menu to define conditions to search for and mark on an I2C bus waveform. You can have multiple searches on the same bus.

**NOTE.** Requires option SREMBD.

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the I <sup>2</sup> C bus that you want to search.
<b>Mark When</b>	Select the type of information for which to search.
<b>Direction</b>	Sets the transfer direction for which to search.  Only available when <b>Mark When = Address or Address &amp; Data</b> .
<b>Addressing Mode</b>	Sets the slave device address length (7 bits or 10 bits long).  Only available when <b>Mark When = Address or Address &amp; Data</b> .
<b>Address</b>	Sets the address pattern for which to search. The number of bits shown depends on the <b>Address Mode</b> setting.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark When = Address or Address &amp; Data</b> .
<b>Data Bytes</b>	Sets the number of data bytes for which to search (one to five bytes). Use the <b>A</b> knob to change the value.  Only available when <b>Mark When = Data or Address &amp; Data</b> .
<b>Data</b>	Sets the data pattern for which to search. The number of bits shown depends on the <b>Data Bytes</b> setting.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark When = Data or Address &amp; Data</b> .
<b>A, B knob controls</b>	Use the <b>A</b> knob to select (highlight) the character to change.  Use the <b>B</b> knob to change the value of the character.
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**I3C serial bus Search configuration menu**

Use the I3C Search configuration menu to define conditions to search for and mark on an I3C bus waveform. You can have multiple searches on the same bus.

To access an I3C serial bus Search menu, double-tap on a I3C Search badge in the Results bar. If there are no I3C Search badges, tap the **Search** button and add an I3C search.

---

**NOTE.** You must have an I3C serial bus waveform in the **Waveform View** before you can create a Search for the bus.

---



---

**NOTE.** Requires option 5/6-SRI3C.

---

**Table 8: I3C search menu, fields and controls**

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks for this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the I3C bus that you want to search.
<b>Mark On</b>	<p>Sets the type of information for which to search. If the selected search type requires more input, the menu updates to show additional fields.</p> <p>See the following <b>Mark On</b> tables for the menu fields associated with specific <b>Mark On</b> settings.</p>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**Mark On = Start.** This table lists the menu fields displayed when **Mark On** is set to **Start**.

Field or control	Description
<b>Start</b>	<p>Sets the search to mark <b>Start</b> events. A <b>Start</b> event occurs when the SDA line transitions from a High to Low state while the SCL line remains High.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>

**Mark On = Repeated Start.** This table lists the menu fields displayed when **Mark On** is set to **Repeated Start**.

Field or control	Description
<b>Repeated Start</b>	<p>Sets the search to mark <b>Repeated Start</b> events. A <b>Repeated Start</b> is when two or more instances of a <b>Start</b> in a row occur without an intervening Stop.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>



**Mark On = Address.** This table lists the menu fields displayed when **Mark On** is set to **Address**.

Field or control	Description
<b>Direction</b>	Sets the search to mark address events of the specified direction ( <b>Read</b> , <b>Write</b> , or <b>Either</b> ).
<b>Addressing Mode</b>	Sets the search to mark <b>7-Bit</b> or <b>10-Bit</b> address events.
<b>Address</b>	<p>Sets the search to mark events that meet the specified address for a Device or register location. The <b>Addressing Mode</b> sets the number of bits in the field.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p>

**Mark On = Data.** This table lists the menu fields displayed when **Mark On** is set to **Data**.

Field or control	Description
<b>Data Bytes</b>	Sets the number of data bytes to show in the <b>Data</b> field. Valid range is 1-5.
<b>Data</b>	<p>Sets the search to mark events that meet the specified data value.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p>

**Mark On = I3C SDR Direct fields.** This table lists the menu fields available in the **I3C SDR Direct** field when **Mark On** is set to **I3C SDR Direct**.

I3C SDR Direct field	Description and menu fields
<b>Enable Slave</b>	<p>Sets the search to mark Enable events (ENEC). Events occur when the Master enables Slave-initiated interrupts on the I3C Bus, such as to indicate a Hot-Join event or request an interrupt.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Disable Slave</b>	<p>Sets the search to mark Disable events (DISEC). Events occur when the Master disables Slave-initiated interrupts on the I3C Bus.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Enter Activity State</b>	<p>Sets the search to mark <b>Enter Activity State</b> events (ENTAS0, ENTAS1, ENTAS2, ENTAS3). Events occur when the Master sets an activity mode state (0, 1, 2, or 3).</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Reset Dynamic Address</b>	<p>Sets the search to mark <b>Reset Dynamic Address</b> events (RSTDAA). Events occur when a Master device tells a Slave device to forget its current Dynamic address and wait for a new address assignment.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Set Max Write Length</b>	<p>Sets the search to mark <b>Set Max Write Length</b> events (SETMWL). Events occur when the Master sets a maximum data write length for a Slave device.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Set Max Read Length</b>	<p>Sets the search to mark <b>Set Max Read Length</b> events (SETMRL). Events occur when the Master sets a maximum data read length for a Slave device.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Set Exchange Time</b>	<p>Sets the search to mark <b>Set Exchange Time</b> events (SETXTIME). Events occur when Master and Slave devices exchange event timing information to collect or reconstruct timestamps, synchronize controls, and do other related tasks.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Set Dynamic Address</b>	<p>Sets the search to mark <b>Set Dynamic Address</b> events (SETDASA). Events occur when the Master assigns a dynamic address to one Slave device using the Slave's static address.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Set New Dynamic Address</b>	<p>Sets the search to mark <b>Set New Dynamic Address</b> events (SETNEWDA). Events occur when the Master assigns a new dynamic address to an I3C Slave device.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>

I3C SDR Direct field	Description and menu fields
<b>Get Max Write Length</b>	<p>Sets the search to mark <b>Get Max Write Length</b> events (GETMWL). Events occur when the Master gets a maximum data write length in bytes for a slave device.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Get Max Read Length</b>	<p>Sets the search to mark <b>Get Max Read Length</b> events (GETMRL). Events occur when the Master gets a maximum data read length in bytes for a slave device.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Get Provisional ID</b>	<p>Sets the search to mark <b>Get Provisional ID</b> events (GETPID). Events occur when the Master sends a Get request to one I3C Slave device to return its 48-bit provisional ID to the Master device.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Get Bus Characteristics</b>	<p>Sets the search to mark <b>Get Bus Characteristics</b> events (GETBCR). Events occur when the Master sends a Get request to one I3C Slave device to return its Bus Characteristics Register (BCR) contents.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Get Dev Characteristics</b>	<p>Sets the search to mark <b>Get Dev Characteristics</b> events (GETDCR). Events occur when the Master sends a Get request to one I3C Slave device to return its Device Characteristics Register (DCR) contents.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Get Slave Current Status</b>	<p>Sets the search to mark get device status events (GETSTATUS).</p> <p>When the <b>Packet</b> button is set to <b>Direct</b>, all events are marked.</p> <p>When the <b>Packet</b> button is set to <b>Response</b>, the menu displays the following fields to refine the search:</p> <p><b>Slave Address</b> sets the Slave's original 7-bit static I<sup>2</sup>C address (or 00 if the slave does not have a static I<sup>2</sup>C address) for which to search.</p> <p><b>Get Status MSB</b> sets the most significant bits of the Slave's status register for which to search. These 8 bits contain vendor-specific information.</p> <p><b>Get Status LSB</b> sets the least significant bits of the Slave's status register for which to search. These 8 bits contain pending interrupt, protocol error, and activity mode bits.</p>
<b>Get Accept Mastership</b>	<p>Sets the search to mark <b>Get Accept Mastership</b> events (GETACCMST). Events occur when a Master request occurs, and when the Current Master offers Mastership to an I3C Secondary Master.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>

I3C SDR Direct field	Description and menu fields
<p><b>Set Bridge Direct Target</b></p>	<p>Sets the search to mark <b>Set Bridge Direct Target</b> events (SETBRGTGT). A Bridge device is a device on the I3C Bus that allows conversion from the native I3C Bus protocol to another protocol such as UART, SPI, and others).</p> <p>When the <b>Packet</b> button is set to <b>Direct</b>, all <b>Set Bridge Direct Target</b> events are marked.</p> <p>When the <b>Packet</b> button is set to <b>Response</b>, the menu displays the following fields to refine the search:</p> <p><b>Slave Address</b> sets the Slave's original 7-bit static I2C address, or 00 if the slave does not have a static I2C address.</p> <p><b>Dynamic Address</b> sets the 7-bit slave device address for which to search.</p> <p><b>ID</b> sets the 16-bit unambiguous identifier for the Bridged device, for which to search.</p>
<p><b>Get Max Data Speed</b></p>	<p>Sets the search to mark <b>Get Max Data Speed</b> events (GETMXDS) for a specific Slave device. Events occur when the Master asks a Slave for its SDR Mode maximum Read and Write data speeds (&amp; optionally max. Read Turnaround time)</p> <p>When the <b>Packet</b> button is set to <b>Direct</b>, all <b>Get Max Data Speed</b> events are marked.</p> <p>When the <b>Packet</b> button is set to <b>Response</b>, the menu displays the following fields to refine the search:</p> <p><b>Slave Address</b> sets the 7-bit address of a specific Slave device for which to search.</p> <p><b>Max Write</b> is an 8-bit field that sets the maximum sustained data write speed (Master to Slave) for which to search.</p> <p><b>Max Read</b> is an 8-bit field specifying the maximum sustained data read speed (Slave to Master) and Clock to Data Turnaround Time for which to search.</p> <p><b>Max Read Turn</b> is a 24-bit field specifying the Maximum Read Turnaround Time for which to search.</p>
<p><b>Get HDR Capability</b></p>	<p>Sets the search to mark <b>Get HDR Capability</b> (GETHDRCAP) events. Events occur when the Master queries a Slave device to determine what HDR Mode(s) that device supports.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>

I3C SDR Direct field	Description and menu fields
<b>Get Exchange Time</b>	<p>Sets the search to mark <b>Get Exchange Time</b> events (GETXTIME).</p> <p>Use the following fields to refine the search:</p> <p>When the <b>Packet</b> button is set to <b>Direct</b>, all <b>Get Exchange Time</b> events are marked.</p> <p>When the <b>Packet</b> button is set to <b>Response</b>, the menu displays the following fields to refine the search:</p> <p><b>Slave Address</b> sets the 7-bit address of a specific Slave device for which to search.</p> <p><b>Supported Modes Byte</b> is an 8-bit field that sets which Timing Control Mode(s) to search for on the specified Slave device.</p> <p><b>State Byte</b> is an 8-bit field that sets the current Timing Control Mode (if any) of the Slave device for which to search, and whether any counter overflows have occurred since the most recent previous check.</p> <p><b>Frequency Byte</b> is an 8-bit field that sets the Slave's internal oscillator frequency for which to search.</p> <p><b>Inaccuracy Byte</b> is an 8-bit field that sets the maximum variation of the Slave's internal oscillator for which to search.</p>

**Mark On = I3C SDR Broadcast fields.** The following table lists the fields available when **Mark On** is set to **I3C SDR Broadcast**.

I3C SDR Broadcast field	Description and menu fields
<b>Enable Slave</b>	<p>Sets the search to mark <b>Enable Slave</b> events (ENEC) by the Master device. Enable Slave controls when Slave-initiated traffic is enabled (allowed) by the Master on the I3C Bus. This function governs a Slave's attempts to request an Interrupt, to request Mastership, or to signify a Hot-Join event.</p> <p>Use the following field to refine the search:</p> <p><b>Event Byte</b> sets the Slave enable value for which to search.</p>
<b>Disable Slave</b>	<p>Sets the search to mark <b>Disable Slave</b> events (DISEC) by the Master device. Disable Slave controls when Slave-initiated traffic is disabled (not allowed) by the Master on the I3C Bus. This function governs a Slave's attempts to request an Interrupt, to request Mastership, or to signify a Hot-Join event.</p> <p>Use the following field to refine the search:</p> <p><b>Event Byte</b> sets the Slave disable value for which to search.</p>
<b>Enter Activity State</b>	<p>Sets the search to mark <b>Enter Activity State</b> events (ENTAS0, ENTAS1, ENTAS2, ENTAS3). Events occur when the Master sets an activity mode state (0, 1, 2, or 3).</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Reset Dynamic Address</b>	<p>Sets the search to mark <b>Reset Dynamic Address</b> events (RSTDAA). Events occur when the Master clears/resets one or all Master-assigned Dynamic addresses from Slave devices.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Enter Dynamic Address</b>	<p>Sets the search to mark <b>Enter Dynamic Address Assignment</b> events (ENTDAA). Events occur when the Master requests Slave devices to enter the Dynamic Address Assignment process.</p> <p>Use the following fields to refine the search:</p> <p><b>Slave Address</b> sets the 7-bit address of a Slave device for which to search.</p> <p><b>ID</b> sets the Slave ID for which to search.</p> <p><b>BCR</b> sets the Bus Control Register values for which to search.</p> <p><b>DCR</b> sets the Device Control Register values for which to search.</p>
<b>Set Max Write Length</b>	<p>Sets the search to mark <b>Set Max Write Length</b> events (SETMWL). Events occur when the Master sets a maximum data write length in bytes for a slave device.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>

I3C SDR Broadcast field	Description and menu fields
<b>Set Max Read Length</b>	<p>Sets the search to mark <b>Set Max Read Length</b> events (SETMRL). Events occur when the Master sets a maximum data read length in bytes for a slave device.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Define List of Slaves</b>	<p>Sets the search to mark <b>Define List of Slaves</b> events (DEFSLVS). Events occur when Secondary Masters are told the address of the current Master and any Slaves that are present on the I3C Bus.</p> <p>Use the following fields to refine the search:</p> <p><b>Dynamic Address</b> sets the devices' dynamic 7-bit address for which to search.</p> <p><b>DCR Type</b> sets the Device Control Register values for which to search.</p> <p><b>BCR Type</b> sets the Bus Control Register values for which to search.</p> <p><b>Static Address</b> sets the devices' 7-bit static address for which to search.</p>
<b>Enter Test Mode</b>	<p>Sets the search to mark <b>Enter Test Mode</b> events (ENTTM). Events occur when all I3C devices are told that the Master is entering a specified Test Mode during manufacturing or Device test.</p> <p>Use the following field to refine the search:</p> <p><b>Test Mode</b> sets the text mode value for which to search.</p>
<b>Set Exchange Time</b>	<p>Sets the search to mark <b>Set Exchange Timing Information</b> events (SETXTIME). Events occur when Master(s) and Slave(s) are directed to exchange event timing information for purposes including synchronizing controls, collecting or reconstructing timestamps, and specifying the timing data procedure.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Set Static as Dynamic Address</b>	<p>Sets the search to mark events where the Master assigns a Dynamic address to a Slave with a known Static address.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>

**Mark On = I3C DDR.** The following table lists the fields available when **Mark On** is set to **I3C DDR**.

I3C DDR field	Description and menu fields
<b>I3C DDR</b>	<p>Sets the search to mark Double Data Rate (DDR) events.</p> <p>Use the following fields to refine the search:</p> <p><b>Slave Address</b> sets the 7-bit address of a Slave device for which to search.</p> <p><b>Command Code</b> sets the read/write values of a command word for which to search.</p> <p><b>Data Words</b> sets the length of the data words for which to search, in the range of integer value 1 (16 bits) to 5 (80 bits).</p> <p><b>Data</b> sets the data value for which to search.</p>

**Mark On = Errors fields.** The following table lists the fields available when **Mark On** is set to **Errors**.

Errors field	Description
<b>Missing Ack</b>	<p>Sets the search to mark missing ACK error events.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>T-Bit</b>	<p>Sets the search to mark <b>T-Bit</b> (transition bit) error events. T-bits are an alternative to the ACK/NACK mechanism.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Parity</b>	<p>Sets the search to mark <b>Parity</b> error events.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Broadcast Address</b>	<p>Sets the search to mark <b>Broadcast Address</b> error events (an error with Broadcast Address commands intended for multiple Slave devices).</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>Preamble</b>	<p>Sets the search to mark <b>Preamble</b> error events. The Preamble is the set of bits preceding the data words in HDR-DDR.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>
<b>CRC5</b>	<p>Sets the search to mark <b>CRC5</b> (Cyclic Redundancy Check, with fifth-order polynomial length) error events.</p> <p>There are no menu fields or controls displayed when this item is selected.</p>



**Mark On = Hot-Join.** The following table lists the fields available when **Mark On** is set to **Hot-Join**.

Hot-Join field	Description and menu fields
<b>Hot-Join</b>	Sets the search to mark <b>Hot-Join</b> events. Hot-Join events allow Slaves to join the I3C Bus after it is already configured.  There are no menu fields or controls displayed when this item is selected.

**Mark On = Direct Message End.** The following table lists the fields available when **Mark On** is set to **Direct Message End**.

Direct Message End field	Description and menu fields
<b>Direct Message End</b>	Sets the search to mark <b>Direct Message End</b> events. These events are end of message notifiers for messages sent to specific Slave devices.  There are no menu fields or controls displayed when this item is selected.

**Mark On = Stop.** The following table lists the fields available when **Mark On** is set to **Stop**.

Stop field	Description and menu fields
<b>Stop</b>	Sets the search to mark <b>Stop</b> events. A Stop event is the I3C Bus condition of a Low to High transition on the SDA line while the SCL line remains High.  There are no menu fields or controls displayed when this item is selected.

**Mark On = HDR Restart.** The following table lists the fields available when **Mark On** is set to **HDR Restart**.

HDR Restart field	Description and menu fields
<b>HDR Restart</b>	Sets the search to mark <b>HDR Restart</b> events. An event occurs when an <b>HDR Restart</b> pattern is sent on the bus.  There are no menu fields or controls displayed when this item is selected.

**Mark On = HDR Exit.** The following table lists the fields available when **Mark On** is set to **HDR Exit**.

HDR Exit field	Description and menu fields
<b>HDR Exit</b>	Sets the search to mark <b>HDR Exit</b> events. An event occurs when an <b>HDR Exit</b> pattern is sent on the bus.  There are no menu fields or controls displayed when this item is selected.

**LIN serial bus search configuration menu**

Use the LIN Search configuration menu to define conditions to search for and mark on an LIN bus waveform. You can have multiple searches on the same bus.

---

**NOTE.** Requires option *SRAUTO*.

---

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the LIN bus that you want to search.
<b>Mark On</b>	Select the type of information for which to search.
<b>Identifier</b>	<p>Sets the identifier pattern for which to search.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Identifier</b> or <b>Identifier &amp; Data</b>.</p>
<b>Data Bytes</b>	<p>Sets the number of data bytes for which to search (one to four bytes). Use the <b>A</b> knob to change the value.</p> <p>Only available when <b>Mark On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>Data</b>	<p>Sets the data pattern for which to search. The number of bits shown depends on the <b>Data Bytes</b> setting.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>Mark When</b>	<p>Sets the mark when condition.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a <b>Data Low</b> and <b>Data High</b> boundary pattern for the specified search range.</p> <p>Only available when <b>Mark On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

### MIL-STD-1553 serial bus search configuration menu

Use the MIL-STD-1553 Search configuration menu to define conditions to search for and mark on an XxMIL-STD-1553X bus waveform. You can have multiple searches on the same bus.

**NOTE.** Requires option SRAERO.

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the MIL-STD-1553 bus that you want to search.
<b>Mark On</b>	Select the type of information for which to search.
<b>Transmit/Receive Bit</b>	Sets the transmit or receive bit state for which to search.  Only available when <b>Mark On = Command</b> .
<b>Mark when RT Address</b>	Sets the RT address condition for which to search.  When set to <b>Inside Range</b> or <b>Outside Range</b> , fields are displayed to set a low and high address for the specified search range.  Only available when <b>Mark On = Command</b> .
<b>Parity</b>	Sets the parity state for which to search.  Only available when <b>Mark On = Command</b> .
<b>Address</b>	Sets the address value for which to search.  Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Command</b> .
<b>Low Address</b>	Sets the low address value for which to search.  Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Command</b> and <b>Mark When RT Address = Inside Range</b> or <b>Outside Range</b>
<b>Subaddress/Mode</b>	Sets the subaddress or mode value for which to search.  Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Command</b> and <b>Mark When RT Address = Inside Range</b> or <b>Outside Range</b>

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

**Parallel bus search configuration menu**

Use the Parallel Search configuration menu to define conditions to search for and mark on an Parallel bus waveform. You can have multiple searches on the same bus.

---

**NOTE.** *Parallel bus search is standard on all instruments.*

---

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the parallel bus that you want to search.
<b>Data</b>	Sets the data pattern for which to search. The number of bits shown depends on how the parallel bus is defined.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
<b>A, B knob controls</b>	Use the <b>A</b> knob to select (highlight) the digit(s) to change.  Use the <b>B</b> knob to change the value of the digit(s).
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**RS-232 serial bus search configuration menu**

Use the RS-232 Search configuration menu to define conditions to search for and mark on an RS-232 bus waveform. You can have multiple searches on the same bus.

---

**NOTE.** *Requires option SRCOMP.*

---

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the RS232 bus that you want to search.
<b>Mark When</b>	Select the type of information for which to search.
<b>Data Bytes</b>	Sets the number of data bytes (1 byte = 8 bits) for which to search (one to ten bytes). Use the <b>A</b> knob to change the value.  Only available when <b>Mark When = Data</b> .

Field or control	Description
<b>Data</b>	<p>Sets the data pattern for which to search. The number of bits shown depends on the <b>Data Words</b> setting.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark When = Data</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

### SENT serial bus search configuration menu

Use the SENT Search configuration menu to define conditions to search for and mark on an SENT bus waveform. You can have multiple searches on the same bus.

---

**NOTE.** Requires option *SRAUTOSEN*.

---

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the <b>SENT</b> bus that you want to search.
<b>Mark On</b>	Select the type of information for which to search.
<b>Channel</b>	<p>Sets the SENT channel type for which to search.</p> <p>Available when <b>Mark On = Start of Packet</b>.</p>
<b>Status / Communications</b>	<p>Sets the value of the status/communications nibble for which to search.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Fast Channel</b>.</p>

Field or control	Description
<b>Fast Channel 1</b>	<p>Sets the condition and value of the fast channel 1 data for which to search.</p> <p>Tap the down arrow and select the condition for which to search (=, ≠, &gt;, &lt;, ≥, ≤). The default is =.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Fast Channel</b>.</p>
<b>Fast Channel 2</b>	<p>Sets the condition and value of the fast channel 2 data for which to search.</p> <p>Tap the down arrow and select the condition for which to search (=, ≠, &gt;, &lt;, ≥, ≤). The default is =.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Fast Channel</b>.</p>
<b>Counter</b>	<p>Sets the condition and value of the counter data for which to search.</p> <p>Tap the down arrow and select the condition for which to search (=, ≠, &gt;, &lt;, ≥, ≤). The default is =.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Fast Channel</b>.</p>
<b>Inverted Nibble</b>	<p>Sets the value of the inverted nibble data for which to search.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Fast Channel</b>.</p>
<b>Message ID</b>	<p>Sets the value of the message ID data for which to search.</p> <p>Available when <b>Mark On = Slow Channel</b>.</p>
<b>Data</b>	<p>Sets the condition and value of the slow channel data for which to search.</p> <p>Tap the down arrow and select the condition for which to search (=, ≠, &gt;, &lt;, ≥, ≤). The default is =.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Slow Channel</b>.</p>

Field or control	Description
<b>Mark When</b>	<p>Sets the mark when condition.</p> <p>Tap the down arrow and select the condition for which to search (=, ≠, &gt;, &lt;, ≥, ≤, <b>Inside Range</b>, <b>Outside Range</b>). The default is =.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set high and low boundary values for the number of clock ticks for which to mark.</p> <p>Available when <b>Mark On = Pause Pulse</b>.</p>
<b>Number of Ticks</b>	<p>Sets the number of Pause Pulse ticks for which to search.</p> <p>Tap the <b>Ticks High</b> or <b>Tick Low</b> field and use the <b>A</b> and <b>B</b> knobs to set the values.</p> <p>Available when <b>Mark On = Pause Pulse</b> and <b>Mark When</b> is set to =, ≠, &gt;, &lt;, ≥, or ≤.</p>
<b>Ticks High, Ticks Low)</b>	<p>Sets the highest and lowest values for the range of Pause Pulse ticks for which to search.</p> <p>Tap the <b>Ticks High</b> or <b>Tick Low</b> field and use the <b>A</b> and <b>B</b> knobs to set the values.</p> <p>Available when <b>Mark On = Pause Pulse</b> and <b>Mark When = Inside Range</b> or <b>Outside Range</b>.</p>
<b>Error Type</b>	<p>Sets the error type for which to search. Tap the arrow and select the error condition.</p> <p>Available when <b>Mark On = Error</b>.</p>
<b>CRC Type</b>	<p>Sets the CRC error type for which to search.</p> <p>Available when <b>Mark On = Error</b> and <b>Error Type = CRC</b>.</p>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.



**SPI serial bus search configuration menu**

Use the SPI Search configuration menu to define conditions to search for and mark on an SPI bus waveform. You can have multiple searches on the same bus.

**NOTE.** Requires option *SREMBD*.

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the SPI bus that you want to search.
<b>Mark On</b>	Select the type of information for which to search.
<b>Data Words</b>	Sets the number of data words (1 word = 8 bits) for which to search (one to sixteen words). Use the <b>A</b> knob to change the value.  Only available when <b>Mark On = Data</b> .
<b>Data</b>	Sets the data pattern for which to search. The number of bits shown depends on the <b>Data Words</b> setting.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Data</b> .
<b>A, B knob controls</b>	Use the <b>A</b> knob to select (highlight) the digit(s) to change.  Use the <b>B</b> knob to change the value of the digit(s).
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**SPMI serial bus search configuration menu**

Use the SPMI Search configuration menu to define conditions for which you want to search and mark on an SPMI bus signal. You can have multiple searches on the same bus.

**NOTE.** Requires option *SRPM*.

Field or control	Description
<b>Display</b>	Enables or disables displaying marks from this search on the bus waveform.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the <b>SPMI</b> bus for which to search.
<b>Mark On</b>	Select the type of information for which to search.

Field or control	Description
<b>Slave Address</b>	<p>Sets the slave address value for which to search.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Reset, Authenticate, Register Read, Register Write, Extended Register Read, Extended Register Write, Ext. Register Read Long, Ext. Register Write Long, DD Block Slave Read, or Register 0 Write.</b></p>
<b>Master Address</b>	<p>Sets the master address value for which to search.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Master Read, Master Write, or DD Block Master Read.</b></p>
<b>Register Address</b>	<p>Sets the register address value for which to search.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Master Read, Master Write, Register Read, Register Write, Extended Register Read, Extended Register Write, Ext. Register Read Long, or Ext. Register Write Long.</b></p>
<b>Data</b>	<p>Sets the data value for which to search.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Mark On = Master Read, Master Write, Register Read, Register Write, Extended Register Read, Extended Register Write, Ext. Register Read Long, Ext. Register Write Long, DD Block Master Read, DD Block Slave Read, or Register 0 Write.</b></p>
<b>Data Bytes</b>	<p>Sets the number of data bytes for which to search. Tap the field and use the <b>A</b> knob to change the value. Or double-tap on the field and use the virtual keypad to enter a value.</p> <p>Available when <b>Mark On = Extended Register Read, Extended Register Write, Ext. Register Read Long, or Ext. Register Write Long.</b></p>
<b>No Response</b>	<p>Sets to search on data that is all zeros (no response). All values in the Data field are set to zero and cannot be edited.</p> <p>Available when <b>Mark On = Master Read, Register Read, Extended Register Read, Ext. Register Read Long, DD Block Master Read, DD Block Slave Read, or Transfer Bus Ownership.</b></p>

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

### USB serial bus search configuration menu

Use the USB Search configuration menu to define conditions to search for and mark on an USB bus waveform. You can have multiple searches on the same bus.

**NOTE.** Requires option *SRUSB2*.

Field or control	Description
<b>Display</b>	Enables or disables displaying search marks on this search.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Source</b>	Select the USB bus that you want to search.
<b>Mark On</b>	Select the type of information for which to search.
<b>Handshake Type</b>	Sets the handshake packet type for which to search.  Only available when <b>Mark On = Handshake Packet</b> .
<b>Packet Type</b>	Sets the special packet type for which to search.  Only available when <b>Mark On = Special Packet</b> .
<b>Error Type</b>	Sets the error type for which to search.  Only available when <b>Mark On = Error</b> .
<b>Address</b>	Sets the token packet address pattern for which to search. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition.  Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Only available when <b>Mark On = Token Packet</b> and <b>Token Type = all</b> except <b>SOF (0101)</b> .
<b>Token Type</b>	Sets the token packet type for which to search.  Only available when <b>Mark On = Token Packet</b> .

Field or control	Description
<b>Endpoint</b>	<p>Sets the token packet endpoint pattern for which to search. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition. Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Token Packet</b> and <b>Token Type = all except SOF (0101)</b>.</p>
<b>Frame Number</b>	<p>Sets the frame number pattern for which to search. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Token Packet</b> and <b>Token Type = SOF (0101)</b>.</p>
<b>Data Packet Type</b>	<p>Sets the data packet type for which to search.</p> <p>Only available when <b>Mark On = Data Packet</b>.</p>
<b>Data Bytes</b>	<p>Sets the number of data bytes for which to search (one to sixteen bytes). Tap the field and use the <b>A</b> knob to change the value.</p> <p>Only available when <b>Mark On = Data Packet</b>.</p>
<b>Byte Offset</b>	<p>Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the <b>A</b> knob to change the value.</p> <p>Only available when <b>Mark On = Data Packet</b>.</p>
<b>Data</b>	<p>Sets the data packet pattern for which to search. The number of bits shown depends on the Data Bytes setting. Use in conjunction with the <b>Mark When</b> field to specify the exact search condition.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Data Packet</b>.</p>
<b>Mark When</b>	<p>Sets the mark when condition.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a high and low boundary pattern for the specified search range.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Only available when <b>Mark On = Token Packet</b> or <b>Data Packet</b>.</p>

Field or control	Description
<b>A, B knob controls</b>	Use the <b>A</b> knob to select (highlight) the digit(s) to change. Use the <b>B</b> knob to change the value of the digit(s).
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

### Edge Search configuration menu

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

To create a new edge search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Edge**.
3. Select the search **Source**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

### Edge Search configuration menu fields and controls.

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>Edge</b> .
<b>Source</b>	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.
<b>Level</b>	Sets the amplitude level that the signal must pass through to be considered a valid transition.
<b>Set to 50%</b>	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as $(\text{Top} + \text{Bottom})/2$ .
<b>Slope</b>	Sets the signal transition direction to detect. (rising, falling, or either direction).
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**Other search types.**

*Bus Search configuration menus* on page 320

*Logic search configuration menu* on page 356

*Pulse Width Search configuration menu* on page 359

*Rise/Fall Time Search configuration menu* on page 361

*Runt Search configuration menu* on page 363

*Setup and Hold Search configuration menu* on page 365

*Timeout Search configuration menu* on page 368

*Window Search configuration menu* on page 369

*DDR Read Search configuration menu (Requires option SUP6-DBDDR3)* on page 372

*DDR Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 374

*DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 376

**Logic search configuration menu**

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

To create a new logic search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Logic**.
3. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

### Logic Search configuration menu fields and controls.

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>Logic</b> .
<b>Use Clock Edge?</b>	<p>Enables or disables finding logic patterns that occur on the specified clock edge.</p> <p>Yes places marks on the clock waveform wherever the logic pattern occurs.</p> <p>No places marks on the input signal waveform(s) wherever the logic pattern occurs.</p>
<b>Logic Pattern: Define Inputs</b>	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 <a href="#">Define Inputs</a> .
<b>Mark When</b>	<p>Defines the waveform logic event to mark, when Use Clock Edge is set to No.</p> <ul style="list-style-type: none"> <li>■ <b>Goes True:</b> All conditions change to a true state.</li> <li>■ <b>Goes False:</b> All conditions change to a false state.</li> <li>■ <b>Is True &gt; Limit:</b> Condition remains true longer than a specified time.</li> <li>■ <b>Is True &lt; Limit:</b> Condition remains true for less than a specified time.</li> <li>■ <b>Is True = Limit:</b> Condition remains true for a specified time (within <math>\pm 5\%</math>).</li> <li>■ <b>Is True <math>\neq</math> Limit:</b> Condition does not remain true for a specified time (within <math>\pm 5\%</math>).</li> </ul>
<b>Clock Source</b>	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform.
<b>Clock Edge</b>	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.
<b>Clock Threshold</b>	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).

Field or control	Description
<b>Define Logic</b>	<p>Sets the logic condition that must occur with all inputs.</p> <ul style="list-style-type: none"> <li>■ <b>AND:</b> All conditions are true.</li> <li>■ <b>OR:</b> Any condition is true.</li> <li>■ <b>NAND:</b> One or more conditions are true.</li> <li>■ <b>NOR:</b> No conditions are true.</li> </ul>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**Other search types.**

*Bus Search configuration menus* on page 320

*Edge Search configuration menu* on page 355

*Pulse Width Search configuration menu* on page 359

*Rise/Fall Time Search configuration menu* on page 361

*Runt Search configuration menu* on page 363

*Setup and Hold Search configuration menu* on page 365

*Timeout Search configuration menu* on page 368

*Window Search configuration menu* on page 369

*DDR Read Search configuration menu (Requires option SUP6-DBDDR3)* on page 372

*DDR Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 374

*DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 376



### 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
<b>Ch(x) (analog channels) or D(x) (digital Channels)</b>	<p>Use to select the signal sources logic condition on which to perform the logic search (<b>High, Low, Don't Care</b>).</p> <p>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.</p> <p>To set the threshold levels for digital channels, double-tap the digital Channel badge to open its configuration menu.</p> <p>Use the <b>Threshold</b> field to set the signal level that must be exceeded for that signal to be true (logical 1).</p>
<b>Set All</b>	Sets all signal sources to detect a logic <b>High, Low, or Don't Care</b> condition.

### Pulse Width Search configuration menu

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

To create a new pulse width search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Pulse Width**.
3. Select the search **Source**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

**Pulse Width Search menu fields and controls. >**

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>Pulse Width</b> .
<b>Source</b>	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.
<b>Mark When</b>	<ul style="list-style-type: none"> <li>■ <b>&lt; Limit:</b> A pulse width is less than the specified time limit.</li> <li>■ <b>&gt; Limit:</b> A pulse width is greater than the specified time limit.</li> <li>■ <b>= Limit:</b> A pulse width is equal to the specified time limit.</li> <li>■ <b>≠ Limit:</b> A pulse width does not equal (is greater than or less than) the specified time limit.</li> <li>■ <b>Inside Range:</b> A pulse width is in the specified time range.</li> <li>■ <b>Outside Range:</b> A pulse width is outside of the specified time range.</li> </ul>
<b>Level</b>	Sets the amplitude level that the signal must pass through to be considered a valid transition.
<b>Set to 50%</b>	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
<b>Time Limit</b>	Sets the time period condition to be met.
<b>High Time Limit</b>	<p>Sets the longest acceptable pulse width time period for the range condition.</p> <p>Only available when Mark When = Inside Range or Outside Range.</p>
<b>Low Time Limit</b>	<p>Sets the shortest acceptable pulse width time period for the range condition.</p> <p>Only available when Mark When = Inside Range or Outside Range.</p>
<b>Polarity</b>	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**Other search types.**

*Bus Search configuration menus* on page 320

*Edge Search configuration menu* on page 355

*Logic search configuration menu* on page 356

*Rise/Fall Time Search configuration menu* on page 361

*Runt Search configuration menu* on page 363

*Setup and Hold Search configuration menu* on page 365

*Timeout Search configuration menu* on page 368

*Window Search configuration menu* on page 369

*DDR Read Search configuration menu (Requires option SUP6-DBDDR3)* on page 372

*DDR Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 374

*DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 376

**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 create a new rise/fall time search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Rise/Fall Time**.
3. Select the search **Source**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

**Rise/Fall Time Search configuration menu fields and controls.**

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>Rise/Fall Time</b> .
<b>Source</b>	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.
<b>Mark When</b>	<ul style="list-style-type: none"> <li>■ <b>&lt; Limit:</b> A signal has a rise/fall time less than the specified time limit.</li> <li>■ <b>&gt; Limit:</b> A signal has a rise/fall time greater than the specified time limit.</li> <li>■ <b>= Limit:</b> A signal has a rise/fall time that is equal to the specified time limit (<math>\pm 5\%</math>).</li> <li>■ <b><math>\neq</math> Limit:</b> A signal has a rise/fall time that does not equal (is greater than or less than) the specified time limit (<math>\pm 5\%</math>).</li> </ul>
<b>Time Limit</b>	Sets the time period condition to be met.
<b>Slope</b>	Sets the signal transition direction to detect. (rising, falling, or either direction).
<b>Upper Threshold</b>	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
<b>Lower Threshold</b>	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**Other search types.**

*Bus Search configuration menus* on page 320

*Edge Search configuration menu* on page 355

*Logic search configuration menu* on page 356

*Pulse Width Search configuration menu* on page 359

*Runt Search configuration menu* on page 363

*Setup and Hold Search configuration menu* on page 365

*Timeout Search configuration menu* on page 368

*Window Search configuration menu* on page 369

*DDR Read Search configuration menu (Requires option SUP6-DBDDR3)* on page 372

*DDR Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 374

*DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 376

**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 create a new runt search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Runt**.
3. Select the search **Source**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

**Runt Search configuration menu fields and controls.**

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>Runt</b> .
<b>Source</b>	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.
<b>Mark When</b>	<ul style="list-style-type: none"> <li>■ <b>Occurs:</b> A runt signal event occurs.</li> <li>■ <b>&lt; Limit:</b> A runt signal event occurs that has a pulse width less than the specified time limit.</li> <li>■ <b>&gt; Limit:</b> A runt signal event occurs that has a pulse width greater than the specified time limit.</li> <li>■ <b>= Limit:</b> A runt signal event occurs that has a pulse width that is equal to the specified time limit (<math>\pm 5\%</math>).</li> <li>■ <b><math>\neq</math> Limit:</b> A runt signal event occurs that has a pulse width that does not equal (is greater than or less than) the specified time limit (<math>\pm 5\%</math>).</li> </ul>
<b>Time Limit</b>	<p>Sets the time period condition to be met.</p> <p>Only available when Mark When = &lt; Limit, &gt; Limit, = Limit, or != Limit.</p>
<b>Polarity</b>	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
<b>Upper Threshold</b>	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
<b>Lower Threshold</b>	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**Other search types.**

*Bus Search configuration menus* on page 320

*Edge Search configuration menu* on page 355

*Logic search configuration menu* on page 356

*Pulse Width Search configuration menu* on page 359

*Rise/Fall Time Search configuration menu* on page 361

*Setup and Hold Search configuration menu* on page 365

*Timeout Search configuration menu* on page 368

*Window Search configuration menu* on page 369

*DDR Read Search configuration menu (Requires option SUP6-DBDDR3)* on page 372

*DDR Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 374

*DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 376

**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 create a new setup and hold search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Setup & Hold**.
3. Select the search **Clock Source**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

**Setup & Hold Search configuration menu fields and controls.**

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>Setup &amp; Hold</b> .
<b>Clock Source</b>	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform.
<b>Clock Level</b>	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).
<b>Clock Edge</b>	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.
<b>Data Sources</b>	Sets the data signal source(s). All selected sources must meet the specified setup and hold times. See <a href="#">Setup and Hold Search - Define Inputs configuration menu</a> on page 367.
<b>Setup Time</b>	Sets the length of time that data signal should be stable and not change before a clock edge occurs.
<b>Hold Time</b>	Sets the length of time that data signal should be stable and not change after a clock edge occurs.
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.



**Other search types.**

*Bus Search configuration menus* on page 320

*Edge Search configuration menu* on page 355

*Logic search configuration menu* on page 356

*Pulse Width Search configuration menu* on page 359

*Rise/Fall Time Search configuration menu* on page 361

*Runt Search configuration menu* on page 363

*Timeout Search configuration menu* on page 368

*Window Search configuration menu* on page 369

*DDR Read Search configuration menu (Requires option SUP6-DBDDR3)* on page 372

*DDR Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 374

*DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 376

**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
<b>Ch(x) (analog channels) or D(x) (digital Channels)</b>	<p>Use to add (<b>Include</b>) or exclude (<b>Don't Include</b>) the data signal(s) from available input channels and waveforms.</p> <p>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.</p> <p>To set the threshold levels for digital channels, double-tap the digital Channel badge to open its configuration menu.</p> <p>For analog channels, use the threshold field to set the data signal level that must be exceeded for the signal transition to be true.</p>
<b>Set All</b>	Use to <b>Include</b> or <b>Don't Include</b> 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 create a new timeout search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Timeout**.
3. Select the search **Source**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

**Timeout Search menu fields and controls.**

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>Timeout</b> .
<b>Source</b>	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.
<b>Mark When</b>	<ul style="list-style-type: none"> <li>■ <b>Stays High:</b> The signal stays above the specified threshold level longer than the specified time.</li> <li>■ <b>Stays Low:</b> The signal stays below the specified threshold level longer than the specified time.</li> <li>■ <b>Either:</b> The signal stays above or below the specified threshold level longer than the specified time.</li> </ul>
<b>Threshold</b>	Sets the amplitude level that the signal must pass through to be considered a valid transition.
<b>Set to 50%</b>	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as $(\text{Top} + \text{Bottom})/2$ .
<b>Time Limit</b>	Sets the time period condition to be met.
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

**Other search types.**

*Bus Search configuration menus* on page 320

*Edge Search configuration menu* on page 355

*Logic search configuration menu* on page 356

*Pulse Width Search configuration menu* on page 359

*Rise/Fall Time Search configuration menu* on page 361

*Runt Search configuration menu* on page 363

*Setup and Hold Search configuration menu* on page 365

*Window Search configuration menu* on page 369

*DDR Read Search configuration menu (Requires option SUP6-DBDDR3)* on page 372

*DDR Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 374

*DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 376

**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 create a new window search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **Window**.
3. Select the search **Source**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

**Window Search menu fields and controls.**

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>Window</b> .
<b>Source</b>	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.
<b>Mark When</b>	<ul style="list-style-type: none"> <li>■ <b>Enters Window:</b> The signal outside a window enters the window defined by the upper and lower threshold settings.</li> <li>■ <b>Exits Window:</b> The signal exits the window defined by the upper and lower threshold settings.</li> <li>■ <b>Inside &gt; Limit:</b> The signal remains inside the window longer than the specified time limit.</li> <li>■ <b>Outside &gt; Limit:</b> The signal remains outside the window longer than the specified time limit.</li> </ul>
<b>Upper Threshold</b>	Sets the amplitude threshold value for the upper edge of the window.
<b>Lower Threshold</b>	Sets the amplitude threshold value for the lower edge of the window.
<b>Time Limit</b>	Sets the time period condition to be met.  Only available when Mark When = Inside > Limit or Outside > Limit.
<b>Threshold Crossing</b>  (Trigger When = Outside > Limit)	<ul style="list-style-type: none"> <li>■ <b>Upper:</b> 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.</li> <li>■ <b>Lower:</b> 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.</li> <li>■ <b>Either:</b> A signal remains outside (above or below) the two threshold levels for longer than the specified time limit before crossing either threshold level.</li> <li>■ <b>None:</b> A signal remains outside the two specified threshold levels for longer than a specified time limit.</li> </ul>

Field or control	Description
<b>Threshold Crossing</b> (Trigger When = Inside > Limit)	<ul style="list-style-type: none"> <li>■ <b>Upper:</b> A signal remains between two thresholds for longer than the specified time limit before crossing through the upper threshold.</li> <li>■ <b>Lower:</b> A signal remains between two thresholds for longer than the specified time limit before crossing through the lower threshold.</li> <li>■ <b>Either:</b> A signal remains between two thresholds for longer than the specified time limit before crossing through either the upper or lower threshold.</li> <li>■ <b>None:</b> A signal remains between two threshold levels for longer than a specified time limit.</li> </ul>
<b>Copy Trigger Settings to Search</b>	Sets the search criteria to match the current oscilloscope trigger settings.
<b>Copy Search Settings to Trigger</b>	Sets the current oscilloscope trigger settings to match the search criteria.

#### Other search types.

*Bus Search configuration menus* on page 320

*Edge Search configuration menu* on page 355

*Logic search configuration menu* on page 356

*Pulse Width Search configuration menu* on page 359

*Rise/Fall Time Search configuration menu* on page 361

*Runt Search configuration menu* on page 363

*Setup and Hold Search configuration menu* on page 365

*Timeout Search configuration menu* on page 368

*DDR Read Search configuration menu (Requires option SUP6-DBDDR3)* on page 372

*DDR Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 374

*DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)* on page 376

**DDR Read Search  
configuration menu  
(Requires option SUP6-  
DBDDR3)**

Use the DDR Read search to search read events on source waveforms.

To create a new DDR Read search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **DDR Read**.
3. Select the search **Sources**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

## DDR Read Search configuration menu fields and controls.

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>DDR Read</b> .
<b>Source</b>	Lists the source channel or waveform to use to search.
<b>Standard</b>	Sets the DDR standard ( <b>DDR3</b> or <b>LPDDR3</b> ).
<b>Data Rate</b>	Sets the data rate options for the selected standard.  Data rate values for the <b>DDR3</b> standard are 800, 1066, 1333, 1600, 1866, 2133, Auto, and Custom. Default is 800.  Data rate values for the <b>LPDDR</b> standard are 333, 800, 1066, 1200, 1333, 1466, 1600, 1866, 2133, Auto, and Custom. Default is 333.
<b>Custom Data Rate</b>	Sets the custom data rate. Default is 800 MT/s. Available only when <b>Data Rate = Custom</b> .
<b>Burst Detection Method</b>	Sets the burst detection method to use for the measurement. <ul style="list-style-type: none"> <li>■ DQ/DQS Phase Alignment</li> <li>■ Chip Select, Latency + DQ/DQS Phase Alignment</li> <li>■ Logic State + Burst latency</li> </ul>
<b>Postamble Length (tCK)</b>	Sets the postamble length value. Default is 500 mtCK.
<b>Preamble Type</b>	Sets the preamble type ( <b>Static</b> or <b>Dynamic</b> ). Default is <b>Static</b> .
<b>DQ/DQS Levels</b>	Sets to <b>Auto</b> or <b>Manual</b> . Default is <b>Auto</b> .
<b>Configure</b>	Sets the DQ/DQS level values.  <i>See <a href="#">DQ/DQS Levels Configure menu</a> on page 378.</i>  Only available when <b>DQ/DQS Levels = Manual</b> .
<b>Source 3 (CS)</b>	Sets the CS (Chip Select) signal source. Default is Ch 4.  Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b> .
<b>CAS Min</b>	Sets the CAS (Column Access Strobe) minimum cycles value. Default value is 2.0 cycles.  Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b> .

Field or control	Description
<b>CAS Max</b>	<p>Sets the CAS maximum cycles value. Default value is 3.0 cycles.</p> <p>Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b>.</p> <hr/> <p><b>NOTE.</b> The CAS maximum value should be greater than or equal to the CAS minimum value. If the min value is equal to the max value, the max value automatically increases.</p>
<b>CS Mode</b>	<p>Sets the CS (Chip Select) to Auto or Manual mode. Default is Auto.</p> <p>Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b>.</p>
<b>CS Level</b>	<p>Sets the CS (Chip Select) level value. Default is 500 mV.</p> <p>Available only when <b>CS Mode = Manual</b>.</p>
<b>CS Active</b>	<p>Sets the CS signal active true state to Low or High. Default is Low.</p> <p>Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b>.</p>
<b>Logic Pattern</b>	<p>Sets the Logic pattern values.</p> <p>See <a href="#">DDR Input Configure menu</a> on page 379.</p> <p>Available only when <b>Burst Detection Method = Logic State + Burst latency</b>.</p>
<b>Burst Latency</b>	<p>Sets the burst latency value. Default is 2.5 Cycles.</p> <p>Available only when <b>Burst Detection Method = Logic State + Burst latency</b>.</p>
<b>Tolerance</b>	<p>Sets the tolerance value. Default is 1 Cycle.</p>
<b>Burst Length</b>	<p>Sets the burst length. Default is 8 UI.</p>

**DDR Write Search configuration menu (Requires option SUP6-DBDDR3)**

Use the DDR Write search to search the write events on source waveforms.

To create a new DDR Write search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **DDR Write**.
3. Select the search **Sources**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.



**DDR Write Search configuration menu fields and controls.**

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>DDR Read</b> .
<b>Source</b>	Lists the source channel or waveform to use to search.
<b>Standard</b>	Sets the DDR standard ( <b>DDR3</b> or <b>LPDDR3</b> ).
<b>Data Rate</b>	Sets the data rate options for the selected standard.  Data rate values for the <b>DDR3</b> standard are 800, 1066, 1333, 1600, 1866, 2133, Auto, and Custom. Default is 800.  Data rate values for the <b>LPDDR</b> standard are 333, 800, 1066, 1200, 1333, 1466, 1600, 1866, 2133, Auto, and Custom. Default is 333.
<b>Custom Data Rate</b>	Sets the custom data rate. Default is 800 MT/s. Available only when <b>Data Rate = Custom</b> .
<b>Burst Detection Method</b>	Sets the burst detection method to use for the measurement. <ul style="list-style-type: none"> <li>■ DQ/DQS Phase Alignment</li> <li>■ Chip Select, Latency + DQ/DQS Phase Alignment</li> <li>■ Logic State + Burst latency</li> </ul>
<b>Postamble Length (tCK)</b>	Sets the postamble length value. Default is 500 mtCK.
<b>Preamble Type</b>	Sets the preamble type ( <b>Static</b> or <b>Dynamic</b> ). Default is <b>Static</b> .
<b>DQ/DQS Levels</b>	Sets to <b>Auto</b> or <b>Manual</b> . Default is <b>Auto</b> .
<b>Configure</b>	Sets the DQ/DQS level values.  See <a href="#">DQ/DQS Levels Configure menu</a> on page 378.  Only available when <b>DQ/DQS Levels = Manual</b> .
<b>Source 3 (CS)</b>	Sets the CS (Chip Select) signal source. Default is Ch 4.  Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b> .
<b>CAS Min</b>	Sets the CAS (Column Access Strobe) minimum cycles value. Default value is 2.0 cycles.  Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b> .

Field or control	Description
<b>CAS Max</b>	<p>Sets the CAS maximum cycles value. Default value is 3.0 cycles.</p> <p>Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b>.</p> <hr/> <p><b>NOTE.</b> The CAS maximum value should be greater than or equal to the CAS minimum value. If the min value is equal to the max value, the max value automatically increases.</p>
<b>CS Mode</b>	<p>Sets the CS (Chip Select) to Auto or Manual mode. Default is Auto.</p> <p>Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b>.</p>
<b>CS Level</b>	<p>Sets the CS (Chip Select) level value. Default is 500 mV.</p> <p>Available only when <b>CS Mode = Manual</b>.</p>
<b>CS Active</b>	<p>Sets the CS signal active true state to Low or High. Default is Low.</p> <p>Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b>.</p>
<b>Logic Pattern</b>	<p>Sets the Logic pattern values.</p> <p>See <a href="#">DDR Input Configure menu</a> on page 379.</p> <p>Available only when <b>Burst Detection Method = Logic State + Burst latency</b>.</p>
<b>Burst Latency</b>	<p>Sets the burst latency value. Default is 2.5 Cycles.</p> <p>Available only when <b>Burst Detection Method = Logic State + Burst latency</b>.</p>
<b>Tolerance</b>	<p>Sets the tolerance value. Default is 1 Cycle.</p>
<b>Burst Length</b>	<p>Sets the burst length. Default is 8 UI.</p>

**DDR Read & Write Search configuration menu (Requires option SUP6-DBDDR3)**

Use the DDR Read & Write search to search both read and write events on source waveforms.

To create a new DDR Read & Write search:

1. Tap **Add New... Search**.
2. Set the **Search Type** to **DDR Read & Write**.
3. Select the search **Sources**.
4. Use the menu fields to set the search parameters.

To change the settings on an existing search, double-tap the search badge and make necessary changes.

## DDR Read &amp; Write Search configuration menu fields and controls.

Field or control	Description
<b>Display</b>	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.
<b>Stop Acquisition if Event Found</b>	Stops input acquisition when the search event occurs. Default is not enabled.
<b>Search Type</b>	Set to <b>DDR Read</b> .
<b>Source</b>	Lists the source channel or waveform to use to search.
<b>Standard</b>	Sets the DDR standard ( <b>DDR3</b> or <b>LPDDR3</b> ).
<b>Data Rate</b>	Sets the data rate options for the selected standard.  Data rate values for the <b>DDR3</b> standard are 800, 1066, 1333, 1600, 1866, 2133, Auto, and Custom. Default is 800.  Data rate values for the <b>LPDDR</b> standard are 333, 800, 1066, 1200, 1333, 1466, 1600, 1866, 2133, Auto, and Custom. Default is 333.
<b>Custom Data Rate</b>	Sets the custom data rate. Default is 800 MT/s. Available only when <b>Data Rate = Custom</b> .
<b>Burst Detection Method</b>	Sets the burst detection method to use for the measurement. <ul style="list-style-type: none"> <li>■ DQ/DQS Phase Alignment</li> <li>■ Chip Select, Latency + DQ/DQS Phase Alignment</li> <li>■ Logic State + Burst latency</li> </ul>
<b>Postamble Length (tCK)</b>	Sets the postamble length value. Default is 500 mtCK.
<b>Preamble Type</b>	Sets the preamble type ( <b>Static</b> or <b>Dynamic</b> ). Default is <b>Static</b> .
<b>DQ/DQS Levels</b>	Sets to <b>Auto</b> or <b>Manual</b> . Default is <b>Auto</b> .
<b>Configure</b>	Sets the DQ/DQS level values.  See <a href="#">DQ/DQS Levels Configure menu</a> on page 378.  Only available when <b>DQ/DQS Levels = Manual</b> .
<b>Source 3 (CS)</b>	Sets the CS (Chip Select) signal source. Default is Ch 4.  Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b> .
<b>CAS Min</b>	Sets the CAS (Column Access Strobe) minimum cycles value. Default value is 2.0 cycles.  Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b> .

Field or control	Description
<b>CAS Max</b>	<p>Sets the CAS maximum cycles value. Default value is 3.0 cycles.</p> <p>Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b>.</p> <hr/> <p><b>NOTE.</b> The CAS maximum value should be greater than or equal to the CAS minimum value. If the min value is equal to the max value, the max value automatically increases.</p>
<b>CS Mode</b>	<p>Sets the CS (Chip Select) to Auto or Manual mode. Default is Auto.</p> <p>Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b>.</p>
<b>CS Level</b>	<p>Sets the CS (Chip Select) level value. Default is 500 mV.</p> <p>Available only when <b>CS Mode = Manual</b>.</p>
<b>CS Active</b>	<p>Sets the CS signal active true state to Low or High. Default is Low.</p> <p>Available only when <b>Burst Detection Method = Chip Select, Latency + DQ/DQS Phase Alignment</b>.</p>
<b>Logic Pattern</b>	<p>Sets the Logic pattern values.</p> <p>See <a href="#">DDR Input Configure menu</a> on page 379.</p> <p>Available only when <b>Burst Detection Method = Logic State + Burst latency</b>.</p>
<b>Burst Latency</b>	<p>Sets the burst latency value. Default is 2.5 Cycles.</p> <p>Available only when <b>Burst Detection Method = Logic State + Burst latency</b>.</p>
<b>Tolerance</b>	<p>Sets the tolerance value. Default is 1 Cycle.</p>
<b>Burst Length</b>	<p>Sets the burst length. Default is 8 UI.</p>

### DQ/DQS Levels Configure menu

Use the DQ/DQS Levels Configure menu to set strobe, data, hysteresis, and margin values.

**DQ/DQS Levels Configure menu.** This menu is accessed from the **Configure** button in a **DDR Read**, **DDR Write**, or **DDR Read & Write** search menu, when **DQ/DQS Levels** is set to **Manual**.

Field or control	Description
<b>High</b>	Sets the high value for Strobe and Data. Default value is 0.
<b>Mid</b>	Sets the mid value for Strobe and Data. Default value is 0.
<b>Low</b>	Sets the low value for Strobe and Data. Default value is 0.
<b>Hysteresis</b>	Sets the hysteresis. Default value is 10%.
<b>Margin</b>	Sets the margin. Default value is 20%.

**DDR Input Configure menu**

Use the Input configure menu to set the logic pattern input values for CS, RAS, CAS, and WE values.

**Logic Pattern Levels Input Configure menu.** This menu is accessed from the **Define Inputs** button in **DDR Read**, **DDR Write**, or **DDR Read & Write** search menu, when **Logic State + Burst Latency** is selected as the **Burst Detection method**.

Field or control	Description
<b>CS</b>	Sets the chip select source, level, and the threshold value.
<b>RAS</b>	Sets the row access strobe source, level, and the threshold value.
<b>CAS</b>	Sets the column access strobe source, level, and the threshold value.
<b>WE</b>	Sets the write enable source, level, and the threshold value.

**Analog Channel configuration menu**

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

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 392.

**Vertical Settings panel, fields and controls**

Field or control	Description
<b>Display</b>	Enables ( <b>On</b> ) or disables ( <b>Off</b> ) displaying the channel waveform in the Waveform View window.
<b>Vertical Scale</b>	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
<b>Offset</b>	Sets the channel signal vertical offset.
<b>Set to 0</b>	Sets the channel signal vertical offset to 0 units.
<b>Position</b>	Double-tap the field to set the vertical position using the virtual keypad.
<b>Set to 0</b>	Tap to set the waveform zero volt level to the center of the slice or waveform view.
<b>Label</b>	Double-tap the field to add a label to the channel display using the virtual keypad.
<b>Coupling</b>	Tap to set the input coupling to DC or AC.
<b>Termination</b>	Tap to set the input termination to 1 M $\Omega$ or 50 $\Omega$ . If you are using a supported TekVPI probe, this value is automatically set by the probe and these controls are not available.

Field or control	Description
<b>Bandwidth Limit</b>	Tap to select the bandwidth limit from the drop-down list.
<b>Bandwidth Filter Optimized For</b>	<p>Tap to select a bandwidth filter that is optimized for flatness or step response. This selection is only available on 6 Series MSO instruments.</p> <p><b>Flatness</b> selects a brick-wall filter optimized for flatness within band with a sharp rolloff. Flatness filtering is not compatible with Peak Detect and Envelope acquisition modes.</p> <p><b>Step Response</b> selects a Bessel-Thompson filter that minimizes overshoot with a gradual rolloff.</p> <p><b>High Res</b> acquisition mode requires Flatness filtering.</p>
<b>Other channel settings panels</b>	
<b>Probe Setup</b>	<p>Use to see probe information, check the probe compensation status, compensate the probe, or restore the factory defaults.</p> <p>See <a href="#">Probe Setup panel (Channel configuration menu)</a> on page 380.</p> <p>See <a href="#">Probe Compensation configuration menu (analog channels Probe Setup panel)</a> on page 382.</p> <p>See <a href="#">Unsupported Probe dialog</a> on page 382.</p>
<b>Spectrum View</b>	<p>Use to enable and configure the spectrum traces to display for the channel.</p> <p>See <a href="#">Spectrum View panel (Channel configuration menu)</a> on page 383.</p>
<b>Other</b>	<p>Use to adjust signal delay to align signal arrival at the oscilloscope between probes and/or cables.</p> <p>See <a href="#">Other panel (Channel configuration menu)</a> on page 385.</p>

**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
<b>Probe Information</b>	View probe information such as probe type, serial number, and its attenuation.
<b>Probe Compensation Status</b>	View the probe compensation status: Default, Pass, or Fail.
<b>Compensate Probe</b>	Tap to display the probe compensation dialog. This is only available for probes that support automatic compensation.
<b>Mode</b>	Select the probe operating mode from the drop down list (Differential, Common Mode, A, or B).
<b>Offset</b>	Enter the A or B offset value. This value is the same as the Offset in the Settings panel.
<b>Differential Offset Tracking</b>	When checked, the Differential Offset Tracking mode is Tracking. When unchecked, the Differential Offset Tracking is Manual.
<b>Differential Offset</b>	When in manual mode, enter the Differential Offset value.
<b>Common Mode Offset</b>	When in manual mode, enter the Common Mode Offset value.
<b>Common Mode Offset Tracking</b>	When checked, the Common Mode Offset Tracking mode is Tracking. When unchecked, the Common Mode Offset Tracking is Manual.
<b>VTerm Tracking</b>	When checked, the VTerm mode is Tracking. When unchecked, the VTerm is Manual.
<b>VTermA</b>	When in manual mode, enter the VTermA value.
<b>VTermB</b>	When in manual mode, enter the VTermB value.

Probe error messages are also displayed in this setup panel, see [Error Messages and Badges](#) on page 67.

**See also.**

[Unsupported Probe dialog](#) on page 382

[Probe Compensation configuration menu \(analog channels Probe Setup panel\)](#) on page 382

[Other panel \(Channel configuration menu\)](#) on page 385

[Deskew configuration menu \(Other panel, Channel configuration menu\)](#) on page 386

**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.**

*Probe Compensation configuration menu (analog channels Probe Setup panel) on page 382*

*Other panel (Channel configuration menu) on page 385*

*Deskew configuration menu (Other panel, Channel configuration menu) on page 386*

**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
<b>Compensate Probe</b>	Tap to compensate the attached probe. Before compensating the probe, read the instructions in the dialog.
<b>Restore Factory Defaults</b>	Tap to restore the probe compensation factory defaults and remove the previous compensation results.
<b>Probe Compensation Status</b>	Probe compensation status can be Running, Passed, Failed or Default.
<b>OK</b>	Tap to close the dialog.



**See also.**

*Unsupported Probe dialog* on page 382

*Other panel (Channel configuration menu)* on page 385

*Deskew configuration menu (Other panel, Channel configuration menu)* on page 386

**Spectrum View panel  
(Channel configuration  
menu)**

Use the Spectrum View panel to enable showing the Spectrum View window, set up the spectrum trace vertical scale units, and select which spectrum traces to display.

To open the **Spectrum View** panel:

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

**Spectrum View panel, fields and controls.**

Field or control	Description
<b>Display</b>	<p>Enables <b>(On)</b> or disables <b>(Off)</b> displaying a spectrum waveform of the channel in a <b>Spectrum View</b> window.</p> <p>If this is the first channel to display a <b>Spectrum View</b>, tapping <b>On</b> opens the <b>Spectrum View</b> window and displays a spectrum trace for the channel. Double-tap other Channel badges to turn on <b>Spectrum View</b> traces for those channels.</p> <p>If this is the last channel to display a <b>Spectrum View</b> trace, tapping <b>Off</b> closes the <b>Spectrum View</b> window.</p> <p>The <b>Spectrum View Display</b> On/Off controls are independent of the <b>Waveform View Display</b> On/Off controls in the <b>Vertical Settings</b> panel.</p>
<b>Units</b>	<p>Sets the measurement unit to use in the vertical scale of this channel in the <b>Spectrum View</b> window. The default value is <b>dBm</b>.</p>
<b>Frequency Domain Traces</b>	<p>Sets the spectrum trace type to display for this channel. You can select to show any combination of traces for each spectrum signal.</p> <p><b>Normal</b> displays the current acquisition, discarding the previous acquisition's data.</p> <p><b>Max Hold</b> displays the maximum data values accumulated over multiple acquisitions of the Normal trace.</p> <p><b>Min Hold</b> displays the minimum data values accumulated over multiple acquisitions of the Normal trace.</p> <p><b>Average</b> displays the average of the Normal trace data over the specified number of acquisitions. This is true power averaging, which occurs before the log conversion. Each power of 2 averaging reduces the displayed noise by 3 dB.</p>
<b>Number of Averages</b>	<p>Sets the number of spectrum acquisitions over which to average the spectrum trace. This control is only shown when <b>Average</b> is selected in the <b>Frequency Domain Traces</b> control.</p>

**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
<b>Deskew</b>	Sets or displays the probe deskew value.
<b>Set to 0</b>	Sets the probe deskew value to zero (0) seconds.
<b>Multi-Channel</b>	Opens a Deskew configuration menu that allows you to deskew multiple channels (two at a time).
<b>External Attenuation</b>	Double-tap the numeric fields to set the external attenuation using the virtual keypad.
<b>Set to Unity</b>	Sets the external attenuation to unity.
<b>Alternate Units</b>	Toggles Alternate Units On or Off. Use to set custom vertical measurement units. The vertical scale will show the entered measurement units.
<b>Units</b>	Double-tap to enter alternate units using the virtual keyboard. Only shown when <b>Alternate Units = On</b> .
<b>Ratio</b>	Double-tap the numeric fields to set the ratio using the virtual keypad. Only shown when <b>Alternate Units = On</b> .
<b>Set to Unity</b>	Sets the ratio to unity. Only shown when <b>Alternate Units = On</b> .

**See also.**

[Unsupported Probe dialog](#) on page 382

[Probe Compensation configuration menu \(analog channels Probe Setup panel\)](#) on page 382

[Deskew configuration menu \(Other panel, Channel configuration menu\)](#) on page 386

**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 94 and [Deskew analog input channels - measurement method](#) on page 95.*

---

**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
<b>From Source</b>	Tap and select from the drop-down list the channel to deskew from (your reference channel for deskewing).
<b>To Source</b>	Tap and select from the drop-down list the channel to deskew to (the channel that you want to match the <b>From Source</b> reference channel).
<b>Probe</b>	<p>If the oscilloscope recognizes the probe attached to the channel, the <b>Probe</b> field shows the nomenclature of the attached probe.</p> <p>If the oscilloscope does not recognize the probe attached to the channel, the <b>Probe</b> field shows a drop-down list from which you can select the probe that is attached to the selected channel.</p> <p>If the attached probe is not in the list, select <b>Custom</b> (at the bottom of the list) and enter the probe propagation delay in the <b>Prop. Delay</b> field.</p>
<b>Prop. Delay</b>	<p>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.</p> <p>This field is available when <b>Probe = Custom</b>.</p>
<b>Propagation Delay</b>	This field lists the default propagation delay of the attached probe. A positive value shifts a channel to the left.
<b>OK, deskew</b>	Sets the oscilloscope to add or subtract the delay values of the <b>To Source</b> channel such that the delay between the two channels is as close to 0 as possible.

**See also.**

*Probe Compensation configuration menu (analog channels Probe Setup panel)* on page 382

*Other panel (Channel configuration menu)* on page 385

*Deskew analog input channels - measurement method* on page 95

*Deskew analog input channels - quick visual method* on page 94

## 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 also select a predefined waveform or load a saved .wfm- or .csv-format waveform from storage (USB drive, or a network drive on instruments with Windows OS installed).

### 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
<b>Output</b>	Tap to toggle the output On or Off.
<b>Waveform Type</b>	Tap to select an available waveform from the list.
<b>Load</b>	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.
<b>Waveform File</b>	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 <b>Load</b> button.
<b>Symmetry</b>	Sets the symmetry of the ramp using the keypad or multipurpose knob.  Only shown when the Waveform Type = Ramp.
<b>Width</b>	Sets the width of the pulse using the keypad or multipurpose knob.  Only shown when the Waveform Type = Pulse.
<b>Duty Cycle</b>	Sets the duty cycle of the square wave using the keypad or multipurpose knob.  Only shown when the Waveform Type = Square.

Field or control	Description
<b>Frequency</b>	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.
<b>Period</b>	Sets the period of the waveform using the keypad or multipurpose knob.
<b>Amplitude</b>	Sets the amplitude of the waveform using the keypad or multipurpose knob
<b>Offset</b>	Sets the offset of the waveform using the keypad or multipurpose knob.
<b>High Level</b>	Sets the High signal amplitude of the waveform using the keypad or multipurpose knob
<b>Low Level</b>	Sets the low signal amplitude of the waveform using the keypad or multipurpose knob.
<b>Load Impedance</b>	Tap to select either <b>50 <math>\Omega</math></b> or <b>High Z</b> (1 M $\Omega$ ) output load impedance.
<b>Add Noise</b>	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 $\Omega$	High Z
<b>Sine</b>	10 mV to 2.5 V	20 mV to 5 V
<b>Square</b>	10 mV to 2.5 V	20 mV to 5 V
<b>Pulse</b>	10 mV to 2.5 V	20 mV to 5 V
<b>Ramp</b>	10 mV to 2.5 V	20 mV to 5 V
<b>DC</b>	0 V to $\pm 1.25$ V	0 V to $\pm 2.5$ V
<b>Noise</b>	10 mV to 2.5 V	20 mV to 5 V
<b>Sine(x)/x</b>	10 mV to 1.5 V	20 mV to 3 V
<b>Gaussian</b>	10 mV to 1.25 V	20 mV to 2.5 V
<b>Lorentz</b>	10 mV to 1.2 V	20 mV to 2.4 V
<b>Exponential Rise</b>	10 mV to 1.25 V	20 mV to 2.5 V
<b>Exponential Decay</b>	10 mV to 1.25 V	20 mV to 2.5 V
<b>Haversine</b>	10 mV to 1.25 V	20 mV to 2.5 V
<b>Cardiac</b>	10 mV to 2.5 V	20 mV to 5 V
<b>Arbitrary</b>	10 mV to 2.5 V	20 mV to 5 V

## Cursor configuration menu

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

To open the Cursors configuration menu:

1. Double-tap a cursor readout or cursor line, or
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
<b>Display</b>	Tap to toggle the cursor display On or Off.
<b>Cursor Type</b>	<p>Tap to select the cursor type from the drop-down list.</p> <p><b>Waveform</b> cursors measure vertical amplitude and horizontal time parameters simultaneously at the point the cursor crosses a waveform.</p> <p><b>V Bars</b> 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.</p> <p><b>H Bars</b> 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.</p> <p><b>V&amp;H Bars</b> cursors measure vertical and horizontal parameters simultaneously. They are not associated with the waveform, but simply show the cursor time and amplitude position.</p>
<b>Source Waveform</b>	<p>Tap to select the source waveform (the waveform on which to display the cursors).</p> <p><b>Same</b> places both cursors on the same waveform.</p> <p><b>Split</b> allows each cursor to be on a different waveform.</p> <p>Only available when Cursor Type = Waveform.</p>



Field or control	Description
<b>Source</b>	Tap to select the source waveform from the drop-down list. <b>Selected Waveform</b> 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.
<b>Cursor A Source, Cursor B Source</b>	Tap to select the waveform sources for Cursor A and B.  Only available when Source Waveform = Split.
<b>Cursor A X-Position</b>	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.
<b>Cursor B X-Position</b>	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.
<b>Cursor Mode</b>	Tap to select the cursor mode.  <b>Independent</b> mode sets multipurpose knobs A and B to move each cursor separately.  <b>Linked</b> 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
<b>Display</b>	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.
<b>Time Format</b>	Tap to select either 12 Hour or 24 Hour time format.
<b>Time Zone</b>	Tap and select the desired time zone from the drop-down list.
<b>Automatically adjust clock for Daylight Saving Time</b>	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 (individually or as a group), 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
<b>Display</b>	Tap to toggle display of the channel On and Off.
<b>Label</b>	Enter label text for the overall FlexChannel data channel.
<b>Height</b>	Sets the relative height of the digital waveform on the screen.  Only available when Waveform View mode = Overlay.
<b>D7-D0 Bit</b>	Tap to toggle individual channels (bits) on or off and remove them from the display.
<b>D7-D0 Threshold</b>	Sets threshold level values for individual data channels.
<b>D7-D0 Label</b>	Enter label text for individual data channels. The label is shown to the right of the corresponding digital channel.
<b>Turn All Off</b>	Turns off the digital channel and removes the Digital channel badge from the bar.
<b>Set All Thresholds</b>	Use to set all data channels to the same threshold value. Enter a value in the field and tap the button.

### Right-click menu differences

A right click (touch and hold) on the digital FlexChannel waveform handle opens a menu to turn off that instrument channel, configure the overall digital channel settings, or add a label to the digital FlexChannel.

A right click (touch and hold) on the handle of individual digital bits in a digital channel waveform opens a menu to turn off that digital bit, configure the overall digital channel settings, or add a label to the individual bit.

**See also** [Analog channel configuration menu](#)

---

## Draw A Box Menu

Use this menu to toggle the Waveform view box draw mode between Zoom area mode and Visual Trigger area mode.

To use the **Draw a Box** menu:

1. Double-tap the **Draw a Box** icon (located at the bottom of the Results Bar).
2. Tap **Zoom** to enable drawing a zoom box area on the screen. Draw a Box mode stays in Zoom mode until changed.
3. Tap **Visual Trigger** to enable drawing Visual Trigger areas on the screen. Draw a Box mode stays in **Visual Trigger** mode until changed.
4. Tap outside the menu. The **Draw a Box icon** changes to reflect the function available when you next select the **Draw a Box** icon.

**See also** [Turn on Zoom mode](#) on page 183  
[Visual Trigger concepts](#) on page 583  
[Create Visual Trigger areas](#) on page 154  
[Edit visual trigger areas on the screen](#) on page 155  
[Edit visual trigger areas using the Area menu](#) on page 158

## 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
<b>Display</b>	Tap to toggle the DVM badge On and Off.
<b>Autorange</b>	Tap to toggle autoranging On and Off. Autorange is not available when the oscilloscope is triggering on the same channel that is being measured.
<b>Source</b>	Tap to select the channel to measure from the drop-down list. The DVM can only measure analog channels.
<b>Mode</b>	Tap to select measuring DC, AC RMS, or DC+AC RMS measurement modes.
<b>Show Basic Statistics in Badge</b>	Tap to toggle showing DVM measurement statistics in the DVM badge.

## Horizontal configuration menu

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

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

**Horizontal configuration menu fields and controls**

Field or control	Description
<b>Horizontal Mode</b>	Tap to select either Automatic or Manual horizontal mode.  In <b>Automatic</b> mode you can set the <b>Minimum Sample Rate</b> and <b>Horizontal Scale</b> .  In <b>Manual</b> mode you can set the Sample Rate and Record Length. Horizontal Scale is a dependent variable calculated from the sample rate and record length.
<b>FastAcq</b>	Tap to toggle <b>FastAcq</b> mode On or Off.  <b>FastAcq</b> 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.

Field or control	Description
<b>FastAcq Palette</b>	<p>Tap and select the desired FastAcq Palette from the drop-down list.</p> <p><b>Temperature</b> 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.</p> <p><b>Spectral</b> 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.</p> <p><b>Normal</b> 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.</p> <p><b>Inverted</b> 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.</p> <p>Only available when FastAcq is On.</p>
<b>Minimum Sample Rate</b>	<p>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.</p> <p>Available when Horizontal Mode = Automatic and Fast Acq = Off.</p> <p>This setting can be overridden if <b>Allow Horizontal Scale to Override Min Sample Rate</b> is selected.</p>
<b>Allow Horizontal Scale to Override Min Sample Rate</b>	<p>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.</p> <p>Only available when Horizontal Mode = Automatic and Fast Acq = Off.</p>
<b>Maximum Record Length</b>	<p>Tap and select the Maximum Record Length from the drop-down list.</p> <p>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.</p> <p>Only available when Horizontal Mode = Automatic and FastAcq = On.</p>
<b>Allow Horizontal Scale to Override Max Record Length</b>	<p>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.</p> <p>Only available when Horizontal Mode = Automatic and FastAcq = On.</p>

Field or control	Description
<b>Horizontal Scale</b>	<p>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.</p> <p>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.</p>
<b>Delay</b>	<p>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).</p>
<b>Position</b>	<p>Tap to set the trigger Position using the assigned multipurpose knob or double-tap to set the Position using the virtual keypad.</p> <p>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.</p> <p>When horizontal delay is off, the trigger point and the horizontal reference are at the same time in the middle of the waveform record.</p>
<b>Set to 0 s</b>	<p>Tap to set the delay position to 0 s (center of the waveform record). Only available when Delay = On.</p>
<b>Set to 10%</b>	<p>Tap to set the trigger delay to 10% of the waveform record.</p> <p>Only available when Delay = Off.</p>
<b>Sample Rate Changes Affect</b>	<p>Tap to allow changes to the Sample Rate to affect either the Horizontal Scale or the Record Length.</p> <p>Only available when Horizontal Mode = Manual.</p>
<b>Sample Rate</b>	<p>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.</p> <p>Only available when Horizontal Mode = Manual.</p>
<b>Record Length</b>	<p>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.</p> <p>Only available when Horizontal Mode = Manual.</p>

## Mask Badge configuration menu

Use the Mask Badge configuration menu to set the total number of eye diagram mask hits needed to fail the mask test.

To open the Mask Badge configuration menu, double-tap a **Mask** badge in the Results column.

### Mask Badge configuration menu fields and controls

Field or control	Description
<b>Total</b>	Sets the total number of hits needed to fail an eye mask test.

## 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 397

[Equation Editor \(Math configuration menu\)](#) on page 401

[Add Filter menu \(math Equation Editor\)](#) on page 402

[Add Variable menu \(math Equation Editor\)](#) on page 403

### 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
<b>Display</b>	Turns the math waveform or FFT plot <b>On</b> or <b>Off</b> .
<b>Vertical Scale</b>	<p>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.</p> <p>Only available when <b>Auto Scale</b> is disabled (unchecked) and <b>Math Type = Basic</b> or <b>Advanced</b>.</p>
<b>Auto Scale</b>	<p>Toggles Auto Scale mode on or off. Auto Scale calculates the vertical scale and position to center and display the entire waveform.</p> <p>Available when <b>Math Type = Basic</b> or <b>Advanced</b>.</p>
<b>Label</b>	Enter a label for the math waveform.
<b>Position</b>	<p>Sets the vertical position of the math waveform.</p> <p>Available when <b>Math Type = Basic</b> or <b>Advanced</b>.</p>
<b>Set to 0</b>	<p>Sets the vertical position of the math waveform to zero (vertical center of a slice (Stacked mode) or the screen (Overlay mode)).</p> <p>Available when <b>Math Type = Basic</b> or <b>Advanced</b>.</p>
<b>Units</b>	Sets user-specified units to display on vertical scale readouts and measurement badges.
<b>Average</b>	Toggles averaging the waveform <b>On</b> and <b>Off</b> . Use averaging to reduce noise on the waveform.
<b>Number of Averages</b>	<p>Sets the number of waveform acquisitions to average. Tap and use the assigned multipurpose knob to set the value.</p> <p>Available when <b>Average = On</b>.</p>



Field or control	Description
<p><b>Math Type</b></p>	<p>Sets the type of math waveform to display.</p> <p><b>Basic</b> creates a math waveform by adding, subtracting, multiplying, or dividing two analog waveforms.</p> <hr/> <p><b>NOTE.</b> You cannot mix time and frequency sources for basic math waveforms. To create a spectrum basic math waveform, both sources must be spectrum waveforms. The spectrum math waveform is added in a new slice in the Spectrum View window.</p> <hr/> <p><b>FFT</b> 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 <a href="#">Math FFT plot configuration menu (Math waveform)</a> on page 450.</p> <hr/> <p><b>NOTE.</b> You cannot create an <b>FFT</b> math waveform from a spectrum trace waveform source.</p> <hr/> <p><b>Advanced</b> 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. See <a href="#">Equation Editor (Math configuration menu)</a> on page 401.</p> <hr/> <p><b>NOTE.</b> You cannot create an advanced math waveform using a spectrum trace waveform source.</p>
<p><b>Source, Source1, Source 2</b></p>	<p>Defines the signal source or sources for a <b>Basic</b> or <b>FFT</b> math waveform.</p> <p>Basic and FFT math waveforms are created from analog channels only (Ch, Math, Ref, or).</p> <p>Available when Math Type = Basic or FFT.</p> <hr/> <p><b>NOTE.</b> You cannot mix time and frequency sources for basic math waveforms. To create a spectrum basic math waveform, both sources must be spectrum waveforms.</p>
<p>Basic math operation list</p>	<p>Located between the <b>Source 1</b> and <b>Source 2</b> fields. A drop-down list to select a basic math operation (add, subtract, multiply, divide) to apply to the two sources.</p> <p>Available when <b>Math Type = Basic</b>.</p>

Field or control	Description
<b>Math n =</b>	<p>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 <b>Equation Editor</b>. Select an equation to display that math waveform.</p> <p>Tap <b>Edit</b> to open the <b>Equation Editor</b> 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 <a href="#">Equation Editor (Math configuration menu)</a> on page 401.</p> <p>Available when <b>Math Type = Advanced</b>.</p>
<b>Edit</b>	<p>Opens the <b>Equation Editor</b> to create advanced math waveforms from analog channels, reference, math waveform, measurement, filter, and variable sources.</p> <p>Tap the <b>Edit</b> button to open the <b>Equation Editor</b>. See <a href="#">Equation Editor (Math configuration menu)</a> on page 401.</p> <p>Available when <b>Math Type = Advanced</b>.</p>

**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 cannot create an **FFT** math waveform from a spectrum trace waveform source.
- You cannot mix time and frequency sources for basic math waveforms. To create a spectrum basic math waveform, both sources must be spectrum waveforms.
- You cannot create an advanced math waveform using a spectrum trace waveform source.
- 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
<b>Sources</b>	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.
<b>Add Filter</b>	Tap to open the <b>Add Filter</b> menu, to create a filter definition to add to the <b>Filters</b> source column in the <b>Equation Editor</b> menu. See <a href="#">Add Filter menu (math Equation Editor)</a> on page 402.
<b>Add Variable</b>	Tap to open the <b>Add Variable</b> menu, to add a defined variable to the <b>Variables</b> source column in the <b>Equation Editor</b> menu. See <a href="#">Add Variable menu (math Equation Editor)</a> on page 403.
<b>Functions</b>	Select the math functions to apply to your signal or signals. See <a href="#">Add Functions (math Equation Editor)</a> on page 403.
<b>Keypad</b>	Use to enter numeric, basic math operations, and logic conditions.
<b>Math &lt;x&gt; =</b>	The field that lists the equation components that you add with the equation controls for the numbered Math waveform. You can also directly edit this field.
<b>Clear</b>	Clears the math equation field.
<b>Arrows</b>	Use to move the cursor position in the <b>Math &lt;x&gt; =</b> equation field.
<b>Delete</b>	Deletes the selected part of the equation (recommend that you use a mouse to more easily select equation text).
<b>Backspace</b>	Deletes the character to the left of the cursor.
<b>Apply</b>	Tap to apply the math expression shown in the <b>Math &lt;x&gt; =</b> field and display the math waveform. The first ~ eight characters of the equation also appears in the Math badge on the Settings bar. When the editor detects a logic or syntax error with the equation, the editor displays an error message, along with the character position in the equation where the editor considers the error to have occurred.
<b>Cancel</b>	Closes the editor menu, does not update the equation list if you made any changes from the last time you Applied or OK'd an equation.
<b>OK</b>	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
<b>Filter Type</b>	<p>Sets the filter type: <b>High Pass</b>, <b>Low Pass</b>, or <b>ArbFlt</b> (arbitrary filter). ArbFlt requires you to load a FLR-format filter file.</p> <p>If loading a filter file, tap Load to navigate to and select the FLR file to load.</p>
<b>Cutoff Frequency</b>	<p>Sets the filter cutoff frequency as a predefined fraction of the sample rate (SR).</p> <p>Default is <math>0.25 * SR</math>.</p>
<b>Label</b>	<p>Shows the filter selections as A:B, where A is the filter type (HP or LP) and B is cutoff frequency setting.</p> <p>Arbitrary filter files use the file name as the label.</p>

**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:




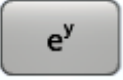
1. Double-tap a Math waveform badge on the Settings bar. If no Math badge is present, tap the **Add New Math** button to add a Math waveform and open the configuration menu.
2. Set Math Type to **Advanced**.
3. Tap **Edit** to open the Equation Editor menu.
4. Tap **Add Variable** to open the Add Variable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

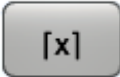


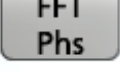



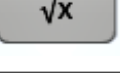
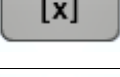

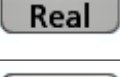
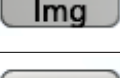
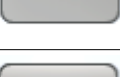

**Add Variable menu fields and controls.**



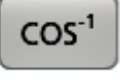


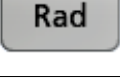
Field or control	Description
<b>Variable Type</b>	Selects a predefined variable to add to the Variables column of the Equation Editor.  <b>Sample Rate</b> creates a variable with the value of the current sample rate.  <b>Sample Interval</b> creates a variable with the value of the current sample interval (1/sample rate)  <b>Record Length</b> creates a variable with the value of the current record length.
<b>Cancel</b>	Closes the menu without adding a variable to the <b>Equation Editor</b> Variables column.
<b>OK</b>	Closes the menu and adds the current variable selection to the Variables column of the <b>Equation Editor's Sources</b> table.

**Add Functions (math Equation Editor)**

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

Button	Description
	Integral. Inserts the text INTG( into the math expression. Enter an argument to the function. The integral function produces the integral of the argument.
	Inserts the text MAX( into the math expression. The MAX function accumulates, over time, the maximum value at each point in the vector.
	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.
	Natural antilog. Inserts the text EXP( into the math expression. The EXP function produces the natural antilog of the argument.

Button	Description
	Ceiling function. Inserts the text CEIL( into the math expression. CEIL returns the smallest integer that is $\geq$ the expression within the parenthesis.
	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 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 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.
	Derivative. Inserts the text DIFF( into the math expression. Enter an argument to the function. The derivative function produces the derivative of the argument.
	Inserts the text MIN( into the math expression. The MIN function accumulates over time the minimum value at each point in the vector.
	Natural logarithm. Inserts the text LN( into the math expression. The natural logarithm function produces the natural logarithm of the argument.
	Square root. Inserts the text SQRT( into the math expression. Enter an argument to the function.
	Floor function. Inserts the text FLOOR( into the math expression. FLOOR returns the largest integer that is $\leq$ the expression within the parenthesis.
	Absolute. Inserts the text FABS( into the math expression. The FABS function takes the absolute value of the expression.
	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 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.
	Inserts the text SIN( into the math expression.
	Inserts the text COS( into the math expression.

Button	Description
	Inserts the text TAN( into the math expression.
	Arc sine. Inserts the text ASIN( into the math expression.
	Arc cosine. Inserts the text ACOS( into the math expression.
	Arc tangent. Inserts the text ATAN( into the math expression.
	Degrees. Inserts the text DEG( into the math expression. The function expresses the value of the expression in degrees.
	Radians. Inserts the text RAD( into the math expression. The function expresses the value of the expression in Radians.

## Menu bar overview

The Menu bar provides access to file, utility, and help functions.

### The Menu bar

Field or control	Description
<b>File</b>	<p>Provides typical system file management operations such as opening, saving, moving, and renaming files.</p> <p>See <a href="#">Recall configuration menu (File menu)</a> on page 407.</p> <p>See <a href="#">Save As configuration menu (File menu)</a> on page 410.</p> <p>See <a href="#">File Utilities configuration (File menu)</a> on page 416.</p> <p><b>Autoset</b> executes an immediate Autoset operation. See <a href="#">Quickly display a waveform (Autoset)</a> on page 100.</p> <p><b>Default Setup</b> immediately restores the oscilloscope to factory default settings. See <a href="#">Using Default Setup</a> on page 132.</p> <p><b>Shutdown</b> powers off the oscilloscope.</p>
<b>Edit</b>	<p>Provides a menu to Undo or Redo the last operation.</p> <p>See <a href="#">Undo, Redo (Edit menu)</a> on page 420.</p>
<b>Utility</b>	<p>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.</p> <p>See <a href="#">User Preferences (Utility menu)</a> on page 420.</p> <p>See <a href="#">I/O (Utility menu)</a> on page 426.</p> <p>See <a href="#">Self Test configuration menu (Utility menu)</a> on page 430.</p> <p>See <a href="#">Calibration configuration menu (Utility menu)</a> on page 431.</p> <p>See <a href="#">Security configuration menu for standard instruments (Utility menu)</a> on page 432.</p>
<b>Application</b>	<p>Use to access installed software applications. This menu only appears when testing applications are installed on the oscilloscope. Only valid for Windows OS installations.</p> <p>See <a href="#">Application (Menu bar)</a> on page 420.</p>
<b>Help</b>	<p>Opens the Help viewer, provides a tutorial to teach touch screen fundamentals, and displays current instrument software and option license information.</p> <p>See <a href="#">Help... (Help menu)</a> on page 438.</p> <p>See <a href="#">User Interface Tutorial (Help menu)</a> on page 438.</p> <p>See <a href="#">About (Help menu)</a> on page 439.</p>



**Recall configuration menu  
(File menu)**

Use this menu to recall (load) reference waveforms, instrument setups, and sessions (setup plus waveforms).

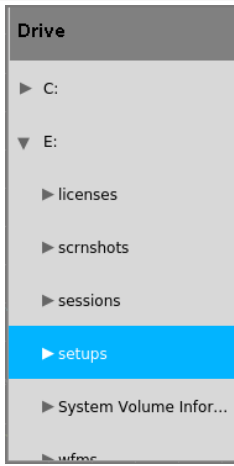


To open the file Recall configuration menu:

1. Tap **File** on the menu bar.
2. Tap **Recall** to open the Recall configuration menu.

**File operations and Microsoft Windows 10 Operating System SSD.** Instruments with 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 Windows 10 installed), which assign a non-changing drive letter to each USB port.

**Recall configuration menu fields and controls.**

Field or control	Description
<b>Look in:</b>	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.
<b>File type to open (tabs)</b>	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.
<b>File name</b>	Lists the selected file name.
<b>Files of type</b>	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.
<b>Cancel</b>	Cancels the file open action and closes the configuration menu.

Field or control	Description
<b>Recall</b>	<p>Recalls the selected file.</p> <p>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.</p> <p>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.</p>

**USB port drive names and locations.** Use the following 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
<b>MSO54, MSO56, MSO58, MSO64, without Windows OS</b>		
<b>Root drive</b>	<b>C</b>	User-accessible memory on the oscilloscope.
<b>Front panel</b>	<b>E</b>	USB 3.0 (left)
	<b>F</b>	USB 2.0 (center)
	<b>G</b>	USB 2.0 (right)
<b>Rear panel</b>	<b>H</b>	USB 2.0 (top)
	<b>I</b>	USB 2.0 (bottom)
	<b>J</b>	USB 3.0 (top)
	<b>K</b>	USB 3.0 (bottom)
<b>MSO58LP, without Windows OS</b>		
<b>Root drive</b>	<b>C</b>	User-accessible memory on the oscilloscope.
<b>Front panel</b>	<b>E</b>	USB 3.0 (left)
	<b>F</b>	USB 2.0 (right)
<b>Rear panel</b>	<b>G</b>	USB 2.0 (top)
	<b>H</b>	USB 2.0 (bottom)
	<b>I</b>	USB 3.0 (top)
	<b>J</b>	USB 3.0 (bottom)
<b>Instruments with Windows OS and USB port labels</b>		
<b>Root drive</b>	<b>C</b>	User-accessible memory on the oscilloscope.
<b>USB ports</b>	<b>Dynamic port letter assignment</b>	<p>If Windows operating system is installed, 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.</p> <p>Use standard Windows procedures to mount and access network drives.</p>

**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 a simple menu selection.*

---

**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 by default; it retains whatever save action was last selected in the **Save As** configuration menu.*

---

**File operations and Microsoft Windows 10 Operating System SSD.** Instruments with 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 Windows 10 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
<b>File save type</b>	Tabs on the left let you set which type of file to save ( <b>Screen Capture</b> , <b>Waveform</b> , <b>Setup</b> , <b>Report</b> , or <b>Session</b> ). Selecting a file type sets the file extensions in the Save As Type field to the correct value.
<b>Save Location</b>	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.
<b>Browse</b>	Tap to open the <b>Browse Save As Location</b> configuration menu, to navigate to and select the location to which to save the file. See <a href="#">Browse Save As Location configuration menu</a> on page 414.
<b>File Name</b>	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.
<b>Auto Increment File Name</b>	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.
<b>Count</b>	Sets the increment count start number. Default is 000.
<b>Save As Type</b>	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.
<b>Cancel</b>	Cancels the file save action and closes the configuration menu.
<b>OK</b>	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
<b>File save type</b>	Tap the <b>Screen Capture</b> 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.
<b>Save As Type</b>	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.

**UtilityUser PreferencesScreen CaptureInverted.Display** *User Preferences (Utility menu)* on page 420

**Waveform tab fields and controls.** The following settings are specific for saving a waveform.

Field or control	Description
<b>File save type</b>	Tap the <b>Waveform</b> tab to save waveform(s) to a file. Selecting <b>Waveform</b> sets the file extensions in the <b>Save As Type</b> field to available waveform file formats.
<b>Save As Type</b>	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.
<b>Source</b>	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.
<b>Gating</b>	Sets the method to save a specified part of the waveform data.  <b>None</b> saves the full waveform data (default).  <b>Cursors</b> saves the waveform data located between the vertical cursors. If cursors aren't on when selecting cursor gating, the cursors are activated.  <b>Screen</b> saves the waveform data that is on the screen.  Gating notes: <ul style="list-style-type: none"> <li>■ Default Setup restores Gated Save to its default setting (None).</li> <li>■ The state of Gated Save gets saved in Setup and Session files.</li> <li>■ Gated saves do not work on plot waveform data.</li> <li>■ Gating cannot be used to save waveform data while in FastFrame mode.</li> </ul>

**Setup tab fields and controls.** The following settings are specific for saving an instrument setup.

Field or control	Description
<b>File save type</b>	Tap the <b>Setup</b> 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.
<b>Include Reference Waveforms</b>	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
<b>File save type</b>	Tap the <b>Report</b> tab to save a report file. Selecting Report sets the file extensions in the Save as Type field to available report file formats.
<b>Save As Type</b>	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).
<b>Append Report</b>	Appends a report to an existing report file.
<b>Comments</b>	Add comments to clarify the contents or purpose of the report, or specifics of the signals being measured.
<b>Include Images and Annotations</b>	Add Waveform and Plot images to the report.
<b>Include Setup Configuration</b>	Add instrument and measurement configuration information 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
<b>File save type</b>	Tap the <b>Session</b> tab to enable saving a session file. Selecting Session sets the file extensions in the Save As Type field to .tss.
<b>Save As Type</b>	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:

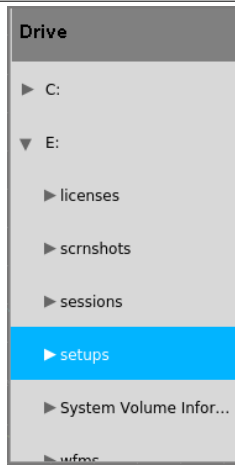


1. Select **File > Save As...**
2. In the Save As configuration menu, tap the **Browse** button.

**File operations and Microsoft Windows 10 Operating System SSD.** Instruments with 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 Windows 10 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
<p><b>File path field</b></p>	<p>Shows the current directory.</p> <p>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.</p> <p>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.</p>
	<p>The Drive column lists the directory structure, opening at the root level. Use to quickly navigate to a location.</p> <p>Tap to list the contents of the directory in the Name pane.</p> <p>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.</p> <p>Drag the list up and down to show more entries.</p>
	<p>Use the arrow buttons to navigate the file directory.</p> <p>The left arrow navigates back to the previously visited folder.</p> <p>The Right arrow navigates forward to the previously visited folder.</p> <p>The Up arrow navigates up one level from the current folder.</p>
	<p>Use to create a new directory (folder) at the current location. Opens the new directory after it is created.</p>
<p><b>Cancel</b></p>	<p>Cancels the file path change, closes the configuration menu, and returns to the Save As configuration menu. No file path change occurs.</p>
<p><b>OK</b></p>	<p>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.</p>

**USB port drive names and locations.**

**File Utilities configuration  
(File menu)**

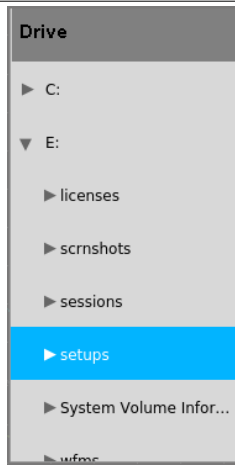


Use this menu to copy, paste, delete, and rename files, and unmount memory devices from USB ports.

To access the **File Utilities** configuration menu, select **File > File Utilities** from the Menu bar.

**File operations and Microsoft Windows 10 Operating System SSD.** Instruments with 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 Windows 10 installed), which assign a non-changing drive letter to each USB port.

**File Utilities configuration menu fields and controls.**

Field or control	Description
<b>File path field</b>	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.
<b>Copy</b>	Copies the selected file in the filename pane to memory.
<b>Paste</b>	Pastes the file from the last Copy action in the current File Utilities session into the current location.
<b>Delete</b>	Deletes the selected file or folder.
<b>Rename</b>	Renames the selected file or folder.
<b>Unmount</b>	Unmount the selected drive. Select the drive letter and tap <b>Unmount</b> .  For USB drives, <b>Unmount</b> closes the file writing session on the attached USB device to let you disconnect the device from the USB port. The device is also removed from the Drive column of menus that can access drives.

**USB port drive names and locations.** Use the following 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
<b>MSO54, MSO56, MSO58, MSO64, without Windows OS</b>		
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)
<b>MSO58LP, without Windows OS</b>		
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)
<b>Instruments with Windows OS and USB port labels</b>		
Root drive	C	User-accessible memory on the oscilloscope.
USB ports	<b>Dynamic port letter assignment</b>	If Windows operating system is installed, 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.

**Mount Network Drive configuration menu**

Use this menu to connect to a network drive. This menu applies to base instruments only (instruments that do not have the Windows 10 OS SSD option installed).

To open the **Mount Network Drive** menu:

1. Select **File > File Utilities** from the Menu bar.
2. Tap **Mount** to open the **Mount Network Drive** menu.

**Mount Network Drive configuration menu fields and controls.**

Field or control	Description
<b>Drive Letter</b>	Shows the current list of available (unassigned) drive letters. Tap on the list and select a drive letter to assign to the network drive.
<b>Specify Server</b>	Sets how you specify the server location; by server <b>Name</b> or <b>IP</b> address.
<b>Server Name</b>	The server name associated with the remote drive. Double-tap the field and enter the server name.
<b>Server IP Address</b>	The IP address of the server. Tap the field and enter the network drive's IP address.
<b>Path</b>	The path to the network drive. Double-tap the field and enter the network drive path information.
<b>User Name</b>	If the drive you are mounting is password-protected, use this field to enter the user name associated with the drive. Double-tap the field and enter the user name.
<b>Password</b>	If the drive you are mounting is password-protected, use this field to enter the password associated with the drive. Double-tap the field and enter the password.
<b>Cancel</b>	Closes the menu without taking any action.
<b>OK</b>	Submits the drive access information to the network drive server. If successful, the menu is dismissed, a confirmation message is displayed, and the drive is added to the <b>Drive</b> column of file menus that access drives.  If not successful, the menu remains on screen and an error message is displayed. Use the error message to resolve the login problem.

**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 (optional).

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, Autoset, measurement, jitter and eye analysis settings, custom waveform colors, and other user preferences.

To open the **User Preferences** menu:

1. Tap **Utility** menu.
2. Tap **User Preferences** to open the configuration menu.
3. Tap a panel.

## Display panel fields and controls.

Field or control	Description
<b>Default Waveform View Display Mode</b>	Sets the default waveform view display mode, either <b>Overlay</b> or <b>Stacked</b> .
<b>Waveform View Graticule</b>	<p>Sets the graticules to be <b>Movable</b> or <b>Fixed</b> for both stacked and overlay modes.</p> <p><b>Movable</b> (default) sets graticules to move along with waveforms, and displays graticule units.</p> <p><b>Fixed</b> sets graticules so that they do not move when you change a waveform's vertical or horizontal position. <b>Fixed</b> graticule mode also does not display the Vertical and Horizontal graticule labels. Use <b>Cursors</b> to take on-screen measurements.</p>
<b>Displayed Colors</b>	<p>Tap and select either <b>Normal</b> or <b>Inverted</b> colors to set how the instrument displays waveforms and plots.</p> <p><b>Normal</b> shows waveforms and plots in color with a black background.</p> <p><b>Inverted</b> makes the waveform background white, with graticule markings in black. <b>Inverted</b> does not change the Menu bar, Results bar, or Settings bar colors. Use this setting to save ink on printed screen captures.</p>
<b>Define Custom Colors</b>	Opens the <a href="#">Define Custom Colors</a> menu. This menu lets you change channel, math, and reference waveform colors for both <b>Normal</b> and <b>Inverted</b> display modes.
<b>Screen Capture Colors</b>	<p>Sets how the instrument saves screen images (<b>Normal</b> or <b>Inverted</b> colors).</p> <p><b>Normal</b> saves waveforms and plots in the colors as shown on the screen.</p> <p><b>Inverted</b> makes the waveform background white, with graticule marking in black, just for screen saves. <b>Inverted</b> does not change the Menu bar, Results bar, or Settings bar colors. Also, if <b>Displayed Colors</b> is set to <b>Inverted</b>, selecting <b>Inverted</b> for screen saves does not invert the colors back to normal for the screen save; to save normal colors for a screen save, select <b>Normal</b>.</p>
<b>Backlight</b>	<p>Sets the intensity of the backlight.</p> <hr/> <p><b>NOTE.</b> Performing an Autoset resets the backlight value to High.</p> <hr/>
<b>Auto-Dim</b>	Select On to automatically dim the screen backlight after a specified time.
<b>Time</b>	<p>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.</p> <p>Only available when <b>Auto-Dim</b> is On.</p>

**Autoset panel fields and controls.**

Field or control	Description
<b>Autoset Adjusts</b>	Selects which controls to change as part of the Autoset operation ( <b>Vertical Settings</b> , <b>Horizontal Settings</b> , <b>Trigger Settings</b> , and <b>Acquisition Settings</b> ). The default is for all adjustments to be enabled.
<b>Autoset in Overlay Display Mode Optimizes</b>	<p>Sets whether to optimize waveform resolution or visibility when doing an Autoset while in the Overlay display mode.</p> <p><b>Resolution</b> uses as much of the ADC's range as possible to provide the best vertical resolution and measurement accuracy, but waveforms will overlap each other.</p> <p><b>Visibility</b> scales and positions waveforms so they are visually separated from each other at the expense of vertical resolution and measurement accuracy.</p>

Autoset guidelines:

- Autoset uses the current trigger source as its reference for various operations such as setting horizontal scale.
- If the current trigger type is **Bus** or **Sequence**, or if trigger type is **Edge** and the trigger source is **AC Line**, Autoset uses the lowest-numbered active channel as its reference.
- Autoset changes the acquisition mode to **Sample**.
- Doing a **Default Setup** operation does not change Autoset panel settings.
- All analysis and measurements abort immediately when an Autoset is started, and then restart upon Autoset completion.



**Measurements panel fields and controls.**

Field or control	Description
<b>Shared Reference Levels</b>	<p>Tap to select either Global or Per Source reference levels.</p> <p><b>Global</b> applies the same reference levels to all measurement sources.</p> <p><b>Per Source</b> allows selection of a different reference levels for all measurement sources.</p>
<b>Calculate Reference Levels</b>	<p>Tap to select how often to calculate reference levels.</p> <p><b>First Acq</b> 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.</p> <p><b>Every Acq</b> calculates the reference levels with every acquisition.</p>
<b>Measurement Interpolation</b>	<p>Interpolation defines how the waveform is drawn between waveform record data points. Tap and select the measurement interpolation method.</p> <p><b>Auto</b> selects the best interpolation method.</p> <p><b>Sin(x)/x</b> interpolation computes record points using a curve fit between the actual samples acquired.</p> <p><b>Linear</b> interpolation computes record points between actual acquired samples by using a straight line fit.</p>
<b>Measurement Annotations</b>	<p>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.</p> <p><b>Auto</b> 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.</p> <p><b>Off</b> turns off display of measurement annotations.</p>

**Jitter and Eye Analysis panel fields and controls.**

Field or control	Description
<b>Calculate Reference Levels</b>	Tap to select calculating the reference levels on the first acquisition or on every acquisition.
<b>Horizontal Measurement Units</b>	Tap to select horizontal measurement units of seconds or unit intervals.
<b>Jitter Separation Model</b>	Tap to select the jitter separation model ( <b>Spectral Only</b> or <b>Spectral + BUJ</b> ).
<b>Lock RJ Value</b>	<p>Selecting <b>Lock RJ Value</b> calculates the measurements at the specified random jitter value. The checkbox is unchecked by default.</p> <p>Selecting the checkbox displays a text box where you can enter the RJ value. The default value is 1 ps.</p> <p><b>NOTE.</b> Lock RJ Value cannot be configured when Jitter Separation Model is Spectral + BUJ.</p>

**Other panel fields and controls.**

Field or control	Description
<b>Right Clicks via Touch</b>	Turns <b>On</b> or <b>Off</b> the ability to use touch and hold method to open right-click menus on badges and other screen items.
<b>Time</b>	Sets the time it takes to respond to a touch and hold before opening a right-click menu.

**Define Custom Colors menu**

Use this menu to change the colors used to display channel, math, and reference waveforms in the Waveform view. You can set custom waveform colors for both Normal and Inverted display modes.

To open the **Define Custom Colors** menu:

1. Tap **Utility** > **User Preferences** to open the configuration menu.
2. Tap **Define Custom Colors** in the **Display** panel.

## Define Custom Colors menu fields and controls.

Field or control	Description
<b>Normal Display Colors</b>	<p>Use this menu to change the waveform colors to use when the instrument is in Normal display mode.</p> <p><b>Channels</b> sets the colors used to display instrument channels.</p> <p><b>Maths</b> sets the colors used to display math waveforms.</p> <p><b>Refs</b> sets the colors used to display reference waveforms.</p> <p>To change a color, tap a channel, math, or reference color to open a color menu and select a new color. The waveform immediately changes to the selected color if the display is in <b>Normal</b> mode. The color menu stays open to let you experiment with different waveform colors. Tap outside the color menu to close it.</p>
<b>Inverted Display Colors</b>	<p>Use this menu to change the waveform colors to use when the instrument is in Inverted display mode.</p> <p><b>Channels</b> sets the colors used to display instrument channels.</p> <p><b>Maths</b> sets the colors used to display math waveforms.</p> <p><b>Refs</b> sets the colors used to display reference waveforms.</p> <p>To change a color, tap a channel, math, or reference color to open a color menu and select a new color. The waveform immediately changes to the selected color if the display is in <b>Inverted</b> mode. The color menu stays open to let you experiment with different waveform colors. Tap outside the color menu to close it.</p> <p>The colors for Math waveforms 1-8 are reused for Math waveforms 9-16, 17-24, and so on.</p> <p>The colors for Ref waveforms 1-8 are reused for Ref waveforms 9-16, 17-24, and so on.</p>
<b>Set All Normal Display Colors to Default</b>	Sets all <b>Normal</b> display mode waveform colors to their factory default settings.
<b>Set All Inverted Display Colors to Default</b>	Sets all <b>Inverted</b> display mode waveform colors to their factory default settings.

### Custom waveform color guidelines.

- There are 48 colors from which to choose.
- There are no restrictions on assigning colors. You can assign different colors to different waveforms, or the same color to many waveforms.
- Changing a waveform color automatically changes all indicators that use that waveform's color (channel handles, vertical scale readout, signal badges, measurement badges, front-panel control LED colors, and so on).
- Waveform color settings are saved as part of waveform, setup, and session save actions.
- You cannot change Bus waveform colors.

### 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
<b>LAN Status</b>	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.
<b>Host Name</b>	The instrument host name is displayed. To change the name, double-tap and enter a name in the virtual keyboard.
<b>Network Address</b>	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.
<b>Domain Name</b>	The instrument domain name is displayed. To change the name, double-tap and enter a name in the virtual keyboard.
<b>Instrument IP Address</b>	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
<b>Subnet Mask</b>	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
<b>Service Name</b>	The instrument service name is displayed. To change the name, double-tap and enter a name in the virtual keyboard.
<b>Gateway IP Address</b>	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
<b>DNS IP Address</b>	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
<b>MAC Address</b>	A readout of the instrument MAC Address. This field is not editable.
<b>e*Scope HTTP Port</b>	A readout of the instrument e*Scope HTTP port number. This field is not editable.
<b>Test Connection</b>	Tap to test the connection. If the connection test is successful, then <b>OK</b> is displayed. If the test is unsuccessful, then <b>No Response</b> is displayed.
<b>LAN Reset</b>	Tap to display the <a href="#">LAN Reset configuration menu (Utility &gt; I/O menu)</a> on page 429.
<b>Apply Changes</b>	Apply changes made on this panel to the instrument.  <b>NOTE.</b> No changes are made to instrument settings until you tap the <i>Apply Changes</i> 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
<b>USB Device Port</b>	Tap to toggle all USB device ports On and Off.
<b>USBTMC Configuration</b>	Displays the USBTMC configuration information.
<b>GPIB Talk/Listen Address</b>	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
<b>Socket Server</b>	Tap to toggle the socket server On or Off.
<b>Protocol</b>	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.
<b>Port</b>	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
<b>AUX Out Signal</b>	Tap to select the signal type to send to the AUX Out connector.  <b>Trigger</b> sends a pulse for each trigger occurrence.  <b>Reference Clock</b> outputs the instrument's reference signal, whether generated internally or supplied from the REF In connector.  <b>AFG Sync</b> sends a pulse that is synchronized to the AFG output signal.
<b>Polarity</b>	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 426

**Self Test configuration menu (Utility menu)**

Use this menu to view power-on diagnostic results, run extended self tests, and verify the 250 kΩ termination control works on input channels.

To open the **Self Test** configuration menu:

1. Tap **Utility** in the Menu bar.
2. Tap **Self Test...**

Tap anywhere outside of the menu to close the menu.

---

**NOTE.** *Remove all input signals before running the extended self tests.*

---



## Self Test configuration menu fields and controls.

Field or control	Description
<b>250 k<math>\Omega</math> Verification</b>	Opens a menu to verify that 250 k $\Omega$ termination can be enabled or disabled for every channel. Closing the menu restores the normal termination setting of all connected probes.
<b>Export Log Files</b>	Opens a menu that lets you save instrument log files to a .zip-compressed file. The log file is a valuable information resource when working with Tektronix Customer Support to troubleshoot a problem or report an issue.  <i><b>NOTE.</b> The <b>Export Log Files</b> button and function is not available on instruments with option <b>5-SEC</b> (Enhanced Security) installed.</i>
<b>Power-on Self test Results</b>	Displays the power-on self test status ( <b>Passed</b> or <b>Failed</b> ).
<b>Extended Self Test Results</b>	Lists the status of each Extended self tests ( <b>Passed</b> or <b>Failed</b> ).  If one or more tests fail at power-on, tap on <b>Run Self Test</b> and see if the test or tests continue to fail. If tests continue to fail, contact your nearest Tektronix Service Center for help with resolving the problem.
<b>Run N Times</b>	Sets the number of times to run the extended self tests.
<b>Run Self Test / Abort Self Test</b>	Tap <b>Run Self Test</b> to run the extended self tests. While tests are running, the button changes to <b>Abort Self Test</b> . When self tests are stopped, the button reverts to <b>Run Self Test</b> .  <i><b>NOTE.</b> Remove all input signals before running the extended self tests.</i>  Tap the <b>Abort Self Test</b> button anytime to stop testing.

**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
<b>Factory Adjustment Status</b>	<p>This area at the top of the menu lists the instrument calibration status. Factory Adjustment Status should be Calibrated.</p> <p>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.</p>
<b>SPC Status</b>	<p>Indicates the status of the last SPC run (Pass or Failed). Also indicates how long ago the last SPC was run.</p>
<b>Run SPC</b>	<p>Signal path Compensation (SPC) corrects for internal DC inaccuracies caused by temperature variations and/or long-term drift in circuits.</p> <hr/> <p><b>NOTE.</b> SPC takes 5 or more minutes per channel to run.</p> <hr/> <p>Allow the instrument to warm up for 20 minutes before running SPC.</p> <p>Remove all probes, cables, and adapters from the input connectors before running SPC.</p> <p>Tap Run SPC to run signal path compensation. The compensation can take up to five minutes per channel.</p>

**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 Enhanced Security when ordered.

---

**NOTE.** This menu applies to all standard instruments that were not ordered with optional Enhanced Security when purchased. For instruments with optional Enhanced Security installed, see [Security configuration menu \(optional\)](#) on page 433.

---

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 (optional)

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 optional Enhanced Security 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 **USB Ports**, **LAN port**, or **Firmware Updates** functions. The default setting is On.

---

**NOTE.** Enabling/Disabling the LAN port is not available on the MSO58LP.

---

4. Tap **Set Password** to enter a password for the first time. If a password has already 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
<b>USB Ports</b>	Tap to enable (On) or disable (Off) all USB ports (Device and Host).
<b>LAN Port</b>	Tap to enable (On) or disable (Off) the Ethernet port.
<b>Firmware Updates</b>	Tap to enable (On) or disable (Off) the ability to update the oscilloscope firmware.
<b>Set Password</b>	Opens the <a href="#">Set Password</a> menu.
<b>Change Password</b>	Opens the <a href="#">Change Password</a> menu.
<b>TekSecure Erase Memory</b>	Tap Run TekSecure to erase nonvolatile memory. It will take approximately seven minutes to erase the memory.
	<b>NOTE.</b> You cannot stop the TekSecure process once it is started.
	<b>NOTE.</b> Do not power down the instrument while TekSecure is running.
	<b>NOTE.</b> Save any important waveform, screen capture, instrument setup, report, and session files to external memory before running TekSecure.
	<b>NOTE.</b> 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 (optional)

Use this function to enter the password to access optional security functions. This menu is only shown on instruments with the optional security function 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
<b>Password</b>	Enter the password. The valid range of characters for the password is from one to 32 characters. Entering no characters or more than 32 characters results in an error message.  <i><b>NOTE.</b> 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.</i>
<b>OK</b>	Closes the menu and toggles the selected function's button when the entered password is correct. Any other conditions result in an error message.
<b>Cancel</b>	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 \(optional\)](#) on page 436.

**Set Password configuration menu (optional)**

Use this function to set the password used to access the optional security functions. This menu is only shown on instruments with optional security functions 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 <sup>1</sup>.**

Field or control	Description
<b>New Password</b>	Enter the new password in this field. <sup>2</sup>
<b>Repeat New Password</b>	Re-enter the new password in this field.
<b>Set New Password</b>	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.
<b>Cancel</b>	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.

**Change Password configuration menu (optional)**

Use this function to change an existing password used to access the optional security functions. This menu is only shown on instruments with optional security functions installed.

To access the Change Password configuration menu:

1. Tap **Utility** in the Menu bar
2. Tap **Security**.
3. Tap **Change Password**.

---

<sup>1</sup> 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.

<sup>2</sup> The valid range of characters for the password is from one to 32 characters. Entering no characters or more than 32 characters results in an error message.

Change Password menu fields and controls<sup>3</sup>.

Field or control	Description
<b>Current Password</b>	Enter the current password in this field. <sup>4</sup>
<b>New Password</b>	Enter the new password in this field. <sup>6</sup>
<b>Repeat New Password</b>	Re-enter the new password in this field. <sup>6</sup>
<b>Delete Password</b>	<p>Deletes the password from the security function. You must enter the correct password before you can clear it.</p> <p>If the password is correct, the password is deleted and the menu closes. The Security menu changes the name of the <b>Delete Password</b> button to <b>Set Password</b>.</p> <p>If the incorrect password is entered, the instrument shows an error message.</p> <hr/> <p><b>NOTE.</b> <i>Once you delete the password, you must set a new password before you can change the security access functions.</i></p>
<b>Set New Password</b>	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.
<b>Cancel</b>	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.

<sup>3</sup> 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.

<sup>4</sup> The valid range of characters for the password is from one to 32 characters. Entering no characters or more than 32 characters results in an error message.

**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.

**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.
<b>Connection Details</b>	Tap this button to open a dialog box that shows how to connect the oscilloscope to a signal source to perform the selected demonstration.
<b>Recall Demo Session</b>	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 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
<b>System information</b>	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.
<b>Probes Detected</b>	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.  <i><b>NOTE.</b> Connecting or disconnecting probes while the About menu is open does not update the Probes Detected list. The Probes Detected list is not dynamic.</i>
<b>Installed Options</b>	Lists installed options. Click on an item in the list to show details of the option in the Option Details area below the list.
<b>Option Details</b>	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.
<b>Remove License</b>	Opens the Location to Save the Exit Key dialog. Use this to uninstall a floating license option for use by others. See <a href="#">Location to Save Exit Key configuration menu</a> on page 439.
<b>Install License</b>	Tap this button to open the Browse License Files dialog to navigate to and select an option license file to install. See <a href="#">Browse License Files menu (Help &gt; About)</a> on page 441.

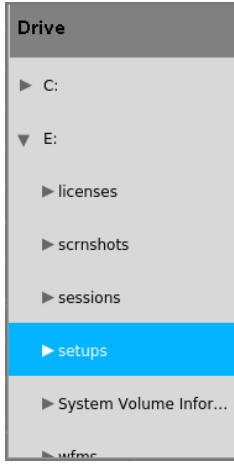


**Location to Save Exit Key configuration menu**

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
<b>Look in</b>	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.
<b>File Name</b>	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.
<b>Files of Type</b>	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.
<b>Cancel</b>	Cancels the license uninstall process, closes the configuration menu without saving any changes that were made, and returns to the About configuration menu.
<b>Create</b>	Closes the configuration menu and saves the license information to the specified location.

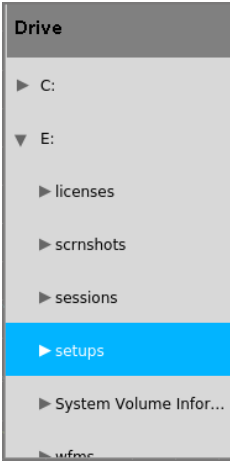


**Browse License Files  
menu (Help > About)**

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
<b>Look in</b>	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.
<b>File Name</b>	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.
<b>Files of Type</b>	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.
<b>Cancel</b>	Cancels the configuration menu changes, closes the menu, and returns to the prior menu without making any changes.

Field or control	Description
<b>Open</b>	Closes the configuration menu, returns to the About configuration menu, and installs the license. Follow any instructions that may be shown during the installation.

## 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
<b>Text</b>	Double-tap and enter the desired text using the virtual keyboard.
<b>Font</b>	Tap and select the desired font from the drop-down list.
<b>Size</b>	Tap and select the desired font size from the drop-down list.
<b>Color</b>	Tap and select the desired font color from the color palette.
<b>B</b>	Tap to toggle text bolding On or Off.
<b>I</b>	Tap to toggle text italics On or Off.
<b>U</b>	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 444

[Math FFT plot configuration menu \(Math waveform\)](#) on page 450

[Histogram plot configuration menu](#) on page 454

[Bathtub plot configuration menu](#) on page 455

[Spectrum plot configuration menu](#) on page 457

[Trend Plot configuration menu](#) on page 459

[Plot XY configuration menu](#) on page 460

[XYZ plot configuration menu](#) on page 461

[Harmonics Bar Graph plot configuration menu \(optional\)](#) on page 462

[SOA plot configuration menu \(optional\)](#) on page 470

[Trajectory plot configuration menu \(Switching Loss power measurement\) \(optional\)](#) on page 471

### 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.

To display an eye diagram plot, see [Add Plot configuration menu](#) on page 306.

**Settings panel (Eye Diagram plot configuration menu) fields and controls.**

Field or control	Description
<b>AutoScale</b>	<p>Toggles <b>AutoScale</b> on or off. Turn AutoScale off to manually set the X and Y-axis range to view an area of interest.</p> <p>When <b>AutoScale</b> is <b>Off</b>, a small Zoom window appears in the plot. Drag the blue vertical bar in the <b>Zoom</b> window to view that area in the main Plot view. Use the <b>Zoom</b> and <b>Pan</b> front-panel knobs to change the zoom area and position.</p>
<b>X-Axis From, To</b>	<p>Sets the beginning and end scale range to display in the plot for the X-Axis scales.</p> <p>Only available when <b>AutoScale</b> is <b>Off</b> (unchecked).</p>
<b>Y-Axis From, To</b>	<p>Sets the beginning and end scale range to display in the plot for the Y-Axis scales.</p> <p>Only available when <b>AutoScale</b> is <b>Off</b> (unchecked).</p>
<b>Gridlines</b>	<p>Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.</p>

**Mask panel (Eye Diagram plot configuration menu) fields and controls.**

Field or control	Description
<b>Mask Test</b>	Toggles mask display on or off.  Setting <b>Mask Test</b> to <b>Off</b> clears and stops mask test calculations.
<b>Mask File</b>	Lists the path and file name of the mask file to load.  You can directly enter the path of the mask file, or use the <b>Browse</b> button to navigate to a mask file. The field also has a drop-down list of recently selected files from which you can also select a mask file to load.  The supported mask file types are: <ul style="list-style-type: none"> <li>■ Legacy base firmware mask files (.msk). Legacy base firmware mask files contain commands that define mask segment geometry and other commands to configure the instrument controls. Only the commands that define the segment geometry are processed.</li> <li>■ DPOJET mask files (.msk). DPOJET mask files use the same format as legacy base firmware mask files but only contain commands that define the segment geometry.</li> <li>■ 5/6 Series MSO mask files (.xml).</li> </ul>
<b>Browse</b>	Opens the <b>Recall Mask File</b> menu that lets you navigate to and select a mask file to load. See <a href="#">Recall Mask File configuration menu</a> on page 447.
<b>Bit Type</b>	Sets which waveform bit types ( <b>Transition</b> , <b>Non-Transition</b> , or <b>All</b> ) to include for eye height analysis.  <b>All</b> does eye analysis using both transition and non-transition bits.  <b>Transition</b> does eye analysis only on transition bits. A transition bit is a bit that is changing from low to high or high to low.  <b>Non-Transition</b> does eye analysis only on nontransition bits. A nontransition bit is a bit that is not changing state.

**Save panel fields and controls.**

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

Field or control	Description
<b>Save Plot Image</b>	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.
<b>Save Plot Data</b>	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.

**Recall Mask File  
configuration menu**

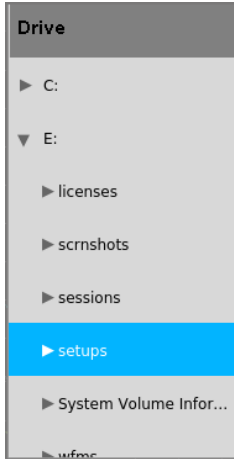


Use this menu to recall (load) an eye diagram mask definition file to apply to an eye diagram plot.

To open the **Recall Mask File** configuration menu:

1. Double-tap anywhere in the **Eye Diagram** plot view to open its configuration menu.
2. Tap the **Mask** panel.
3. Tap **Browse**.

To display an eye diagram plot, see [Add Plot configuration menu](#) on page 306.

**Browse Mask File Location configuration menu fields and controls.**

Field or control	Description
<b>Look in:</b>	Shows the current directory path to the location of a file.  Tap on the file path and use a keyboard to enter the location 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 recalled mask file locations, up to a maximum of 20 locations.
	The <b>Drive</b> column lists the accessible directories, opening at the root (/) level. Use to quickly navigate to a location.  Tap a name in the <b>Drive</b> list to show 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.
	Navigation buttons. 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.
	Create new Directory icon. Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
<b>File name</b>	Shows the selected file path and name.
<b>Files of type</b>	Use to select the file format you want to recall. The drop-down list shows all file extension types that the instrument can read for the selected file type.
<b>Cancel</b>	Cancels the file open action and closes the menu.
<b>Open</b>	Opens (loads) the selected file.  Recalling a mask file adds a Mask badge to the Settings bar and displays the masks in the eye diagram 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
<b>AutoScale</b>	<p>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.</p> <p>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.</p>
<b>Gridlines</b>	<p>Selects which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.</p>
<b>X-Axis From, To</b>	<p>Sets the beginning and end scale range to display in the plot for the X-Axis scales.</p> <p>These values also define the horizontal area shown in the small Zoom window.</p> <p>Available when AutoScale is Off.</p>
<b>Y-Axis From, To</b>	<p>Sets the beginning and end scale range to display in the plot for the Y-Axis scales.</p> <p>These values also define the vertical area shown in the small Zoom window.</p> <p>Available when AutoScale is Off.</p>

#### Save panel fields and controls.

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

Field or control	Description
<b>Save Plot Image</b>	<p>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).</p> <p>Tap OK to save the plot image.</p>
<b>Save Plot Data</b>	<p>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.</p> <p>Tap OK to save the plot data as a comma-separated values (.csv) file.</p>

**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
<b>Auto Scale</b>	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.
<b>Gridlines</b>	Sets which gridlines to show in the plot (Horizontal, Vertical, or Both).
<b>X-Axis Scale</b>	Sets the horizontal frequency scale to either Log or Linear.  A <b>Log</b> 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 <b>Linear</b> scale is useful when the frequency component magnitudes are all close in value, allowing direct comparison of their magnitudes.
<b>Y-Axis Scale</b>	Sets the vertical amplitude scale to either Log (dBm) or Linear.  A <b>Log dB</b> 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 <b>Linear</b> scale is useful when the frequency component magnitudes are all close in value, enabling direct comparison of their magnitudes.
<b>X-Axis From, To</b>	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.
<b>Y-Axis From, To</b>	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
<b>FFT Type</b>	<p><b>Magnitude</b> plots the magnitude values of the frequency components.</p> <p><b>Phase</b> plots the phase of the signal as a function of frequency.</p>
<b>Window</b>	Sets the FFT window type to use for the waveform plot. See <a href="#">FFT windows</a> on page 453.
<b>Gating</b>	<p>Sets the region of the waveform to analyze for the FFT plot.</p> <p><b>None</b> uses the entire waveform record to create the FFT plot.</p> <p><b>Screen</b> uses the part of the waveform record displayed on the screen (such as in Zoom mode) to create the FFT plot.</p> <p><b>Cursors</b> uses the waveform data between the cursors to create the FFT plot.</p>
<b>Vertical Units</b>	<p>Sets the vertical scale to Degrees, Radians, or Group Delay.</p> <hr/> <p><b>NOTE.</b> The Vertical Units set when FFT Type = Phase override the Y-Axis scale setting in the Plot Settings panel.</p> <hr/> <p>Only available when FTT Type set to Phase.</p>
<b>Phase Wrap</b>	<p>When checked, the phase trace is unwrapped where the trace jumps more than the number of degrees set in the adjacent field.</p> <p>Only available when FTT Type set to Phase.</p>
<b>Squelch</b>	<p>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.</p> <p>When unchecked, the phase trace includes all values.</p> <p>Only available when FTT Type set to Phase.</p>

**Save panel (Math FFT plot configuration menu) fields and controls.**

Field or control	Description
<b>Save Plot Image</b>	Opens the <a href="#">Save As</a> configuration menu to specify the location and name at which to save an image file of the Plot view.
<b>Save Plot Data</b>	Opens the <a href="#">Save As</a> 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
<b>Hanning</b>	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.
<b>Rectangular</b>	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.
<b>Hamming</b>	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.
<b>Blackman-Harris</b>	Best magnitude, worst frequency resolution.	Predominantly single frequency signals to look for higher order harmonics.
<b>Kaiser-Bessel</b>	Less spectral leakage than the Hanning, Hamming, or Rectangular windows.	Predominantly single frequency signals to look for higher order harmonics.
<b>Gaussian</b>	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.
<b>FlatTop2</b>	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.
<b>TekExp</b>	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
<b>AutoScale</b>	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.
<b>Gridlines</b>	Selects which gridlines to show in the plot.
<b>X-Axis Number of Bins</b>	Sets the resolution by the number of bins into which the X axis is divided.
<b>X-Axis From, To</b>	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.
<b>Y-Axis From, To</b>	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.
<b>Y-Axis Scale</b>	Sets the Y axis scale to Linear or Log.  A <b>Log</b> 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 <b>Linear</b> 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 data to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
<b>Save Plot Image</b>	<p>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).</p> <p>Tap OK to save the plot image.</p>
<b>Save Plot Data</b>	<p>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.</p> <p>Tap OK to save the plot data as a comma-separated values (.csv) file.</p>

**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
<b>Autoscale</b>	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.
<b>Gridlines</b>	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
<b>X-Axis Units</b>	Sets the X-axis units to be unit intervals or seconds.
<b>X-Axis From, To</b>	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.
<b>Y-Axis From, To</b>	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.
<b>Y-Axis scale</b>	Sets the Y axis scale to Linear (default) or Log.  A <b>Log</b> 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 <b>Linear</b> 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 data to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
<b>Save Plot Image</b>	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.
<b>Save Plot Data</b>	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
<b>AutoScale</b>	<p>Toggles Autoscale on or off. Turn AutoScale off to set the X and Y-axis range to view an area of interest.</p> <p>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.</p>
<b>Gridlines</b>	<p>Sets which gridlines to show in the plot. Use the grid style best suited to your measurement needs.</p>
<b>X-Axis Scale</b>	<p>Sets the X axis scale to Linear or Log.</p> <p>A <b>Log</b> 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.</p> <p>A <b>Linear</b> scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.</p>
<b>Y-Axis Scale</b>	<p>Sets the Y axis scale to Linear or Log.</p> <p>A <b>Log</b> 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.</p> <p>A <b>Linear</b> scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.</p>
<b>Dynamic Range</b>	<p>Sets the vertical scale dynamic range.</p> <p>Available when Y-Axis Scale is set to Log.</p>
<b>X-Axis From, To</b>	<p>Sets the beginning and end scale range to display in the plot for the X-Axis scales.</p> <p>These values also define the horizontal area shown in the small Zoom window.</p> <p>Available when AutoScale is off.</p>
<b>Y-Axis From, To</b>	<p>Sets the beginning and end scale range to display in the plot for the Y-Axis scales.</p> <p>These values also define the vertical area shown in the small Zoom window.</p> <p>Available when AutoScale is off.</p>

**Save panel fields and controls.**

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

Field or control	Description
<b>Save Plot Image</b>	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.
<b>Save Plot Data</b>	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 the trend plot 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.

---

**NOTE.** *Measurements that can use the trend plot as a source will list the plot in the Source menu list.*

---

**Trend configuration menu fields and controls.**

Field or control	Description
<b>Display</b>	Toggles displaying the Trend plot <b>On</b> or <b>Off</b> .
<b>Vertical Scale</b>	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.
<b>Auto Scale</b>	Enables or disables the AutoScale mode. Auto Scale uses the signal data to dynamically set the vertical scale units.
<b>Label</b>	Adds a label to the Trend waveform (appears next to the Trend plot handle).
<b>Position</b>	Shows the vertical position setting (when Auto Scale is on), or sets the vertical position (when Auto Scale is off).
<b>Set to 0</b>	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
<b>X-Axis</b>	Sets the source for the X-axis signal.
<b>Y-Axis</b>	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 data to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
<b>Save Plot Image</b>	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.
<b>Save Plot Data</b>	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
<b>X-Axis</b>	Sets the source for the X-axis signal.
<b>Y-Axis</b>	Sets the source for the Y-axis signal.
<b>Z-Axis</b>	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 data to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
<b>Save Plot Image</b>	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.
<b>Save Plot Data</b>	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 (optional)**

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
<b>AutoScale</b>	Toggles <b>AutoScaleOn</b> or <b>Off</b> . Turn <b>AutoScale</b> off to manually set the X and Y-axis range to view an area of interest.  When <b>AutoScale</b> is <b>Off</b> , 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 <b>Zoom</b> and <b>Pan</b> front-panel knobs to change the zoom area and position.
<b>Gridlines</b>	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
<b>Unit panel</b>	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.
<b>Harmonics</b> (Inside the Unit panel)	Select All/Odd/Even.

**Save panel fields and controls.** Use the Save panel controls to save the plot image or data to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
<b>Save Plot Image</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot image. Enter a file name in the <b>File Name</b> field. Select the image file format (PNG, BMP, or JPG).  Tap <b>OK</b> to save the plot image.
<b>Save Plot Data</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot data. Enter a file name in the <b>File Name</b> field.  Tap <b>OK</b> 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.



**Inductance Curve configuration menu (Magnetic Analysis power measurement) (optional)**

Use this menu to change settings of a displayed Inductance curve plot (Inductance measurement).

To open the Inductance Curve plot configuration menu, double-tap anywhere in an **Inductance Curve Plot** view.

**Settings panel (Inductance Curve configuration menu) fields and controls.**

Field or control	Description
<b>AutoScale</b>	Toggles <b>AutoScale On</b> or <b>Off</b> . Turn <b>AutoScale</b> off to manually set the X and Y-axis range to view an area of interest.  When <b>AutoScale</b> is <b>Off</b> , 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 <b>Zoom</b> and <b>Pan</b> front-panel knobs to change the zoom area and position.
<b>Gridlines</b>	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
<b>Display</b>	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
<b>X-Axis</b>	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed when <b>AutoScale</b> is <b>Off</b> (deselected).
<b>Y-Axis</b>	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed when <b>AutoScale</b> is <b>Off</b> (deselected).

**Save panel fields and controls.** Use the Save panel controls to save the plot image or data to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
<b>Save Plot Image</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot image. Enter a file name in the <b>File Name</b> field. Select the image file format (PNG, BMP, or JPG).  Tap <b>OK</b> to save the plot image.
<b>Save Plot Data</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot data. Enter a file name in the <b>File Name</b> field.  Tap <b>OK</b> 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.

**B-H curve configuration menu (Magnetic Analysis power measurement) (optional)**

Use this menu to change the settings of a displayed B-H curve plot (Magnetic Property measurement).

To open the B-H curve plot configuration menu, double-tap anywhere in a **B-H curve** Plot view.

**Settings panel (B-H curve configuration menu) fields and controls.**

Field or control	Description
<b>AutoScale</b>	Toggles <b>AutoScaleOn</b> or <b>Off</b> . Turn <b>AutoScale</b> of to manually set the X and Y-axis range to view an area of interest.  When <b>AutoScale</b> 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 <b>Zoom</b> and <b>Pan</b> front-panel knobs to change the zoom area and position.
<b>Gridlines</b>	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
<b>Display</b>	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
<b>X-Axis</b>	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed only when <b>AutoScale</b> is Off (deselected).
<b>Y-Axis</b>	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when <b>AutoScale</b> is Off (deselected).
<b>Displayed Cycles</b>	Select either <b>All</b> or <b>B<sub>Peak</sub></b> .

**NOTE.**

- *You can move the cursor symbols in the B-H curve plot by placing waveform cursors at the correct positions in Waveform view. However, to place cursor symbols correctly at exact positions in Waveform view with precision, you need to enable zoom scale on the Waveform view and increase the zoom scale if you need higher resolution, and then move the waveform cursors using MPH knob in fine mode.*
- *When you configure a B-H plot to Bpeak from All cycles, the waveform cursors does not get placed automatically on the peak cycle in the time domain waveform. You have to manually move the waveform cursors until you view the cursor symbols value in the Bpeak plot. This cycle relates to Bpeak.*

**Save panel fields and controls.** Use the Save panel controls to save the plot image or data to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
<b>Save Plot Image</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot image. Enter a file name in the <b>File Name</b> field. Select the image file format (PNG, BMP, or JPG).  Tap <b>OK</b> to save the plot image.
<b>Save Plot Data</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot data. Enter a file name in the <b>File Name</b> field.  Tap <b>OK</b> 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.

**I vs (integral of) V plot configuration menu (Magnetic Analysis power measurement) (optional)**

Use this menu to change settings of a displayed I vs.  $\int V$  plot (I vs.  $\int V$  measurement).

To open the I vs.  $\int V$  plot configuration menu, double-tap anywhere in the **I vs.  $\int V$**  Plot view.

**Settings panel (I vs.  $\int V$  plot configuration menu) fields and controls.**

Field or control	Description
<b>AutoScale</b>	Toggles <b>AutoScaleOn</b> or <b>Off</b> . Turn <b>AutoScale</b> off to manually set the X and Y-axis range to view an area of interest.  When <b>AutoScale</b> 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 <b>Zoom</b> and <b>Pan</b> front-panel knobs to change the zoom area and position.
<b>Gridlines</b>	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
<b>Display</b>	Vectors draws waveforms with lines between record points. Dots draws waveform record points as dots on the screen.
<b>X-Axis</b>	Sets the beginning and end scale range to display in the plot for the X-Axis scales. It is displayed only when <b>AutoScale</b> is off (deselected).
<b>Y-Axis</b>	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. It is displayed only when <b>AutoScale</b> is off (deselected).

**Save panel fields and controls.** Use the Save panel controls to save the plot image or data to a file, for inclusion in reports or further analysis in other applications.

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

**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.

**Control Loop Response  
(Bode) plot configuration  
menu**

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

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

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

Field or control	Description
<b>AutoScale</b>	Toggles <b>AutoScaleOn</b> or <b>Off</b> . Turn <b>AutoScale</b> of to manually set the X and Y-axis range to view an area of interest.  When <b>AutoScale</b> 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 <b>Zoom</b> and <b>Pan</b> front-panel knobs to change the zoom area and position.
<b>Gridlines</b>	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
<b>Display</b>	<b>Vectors</b> draws waveforms with lines between record points. <b>Dots</b> draws waveform record points as dots on the screen.
<b>X-Axis (Frequency) Scale</b>	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed only when <b>AutoScale</b> is Off (deselected).
<b>Y-Axis (Gain) Scale</b>	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when <b>AutoScale</b> is Off (deselected).
<b>Y-Axis (Phase) Scale</b>	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when <b>AutoScale</b> is Off (deselected).

**Save panel fields and controls.** Use the Save panel controls to save the plot image or data to a file, for inclusion in reports or further analysis in other applications.

Field or control	Description
<b>Save Plot Image</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot image. Enter a file name in the <b>File Name</b> field. Select the image file format (PNG, BMP, or JPG).  Tap <b>OK</b> to save the plot image.
<b>Save Plot Data</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot data. Enter a file name in the <b>File Name</b> field.  Tap <b>OK</b> 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.

**Power Supply Rejection Ratio plot configuration menu**

Use this menu to change the settings of a displayed Power Supply Rejection Ratio plot (Power Supply Rejection Ratio measurement).

To open the Power Supply Rejection Ratio plot configuration menu, double-tap anywhere in a **Power Supply Rejection Ratio** Plot view.

**Settings panel (Power Supply Rejection Ratio plot configuration menu) fields and controls.**

Field or control	Description
<b>AutoScale</b>	Toggles <b>AutoScaleOn</b> or <b>Off</b> . Turn <b>AutoScale</b> of to manually set the X and Y-axis range to view an area of interest.  When <b>AutoScale</b> 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 <b>Zoom</b> and <b>Pan</b> front-panel knobs to change the zoom area and position.
<b>Gridlines</b>	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
<b>Display</b>	<b>Vectors</b> draws waveforms with lines between record points. <b>Dots</b> draws waveform record points as dots on the screen.
<b>X-Axis</b>	Sets the beginning and end scale range to display in the plot for the X-Axis scales. This is displayed only when <b>AutoScale</b> is Off (deselected).
<b>Y-Axis</b>	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. This is displayed only when <b>AutoScale</b> is Off (deselected).

**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
<b>Save Plot Image</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot image. Enter a file name in the <b>File Name</b> field. Select the image file format (PNG, BMP, or JPG).  Tap <b>OK</b> to save the plot image.
<b>Save Plot Data</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot data. Enter a file name in the <b>File Name</b> field.  Tap <b>OK</b> 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.

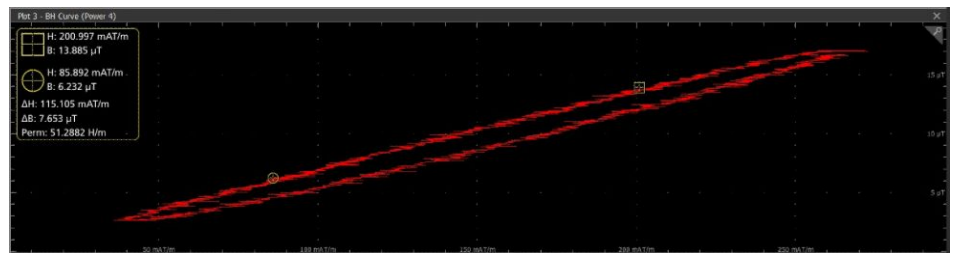
## Power plots and cursors

Use cursors in I vs.  $\int V$  and B-H curve plots to take measurements at any point on these power measurement plot waveforms. Displaying I vs.  $\int V$  and B-H curve power plots is optional.

Cursor readouts display the voltage and current values at their position and the difference (delta) between the cursors. You can display cursors in I vs.  $\int V$  and B-H curve plots.

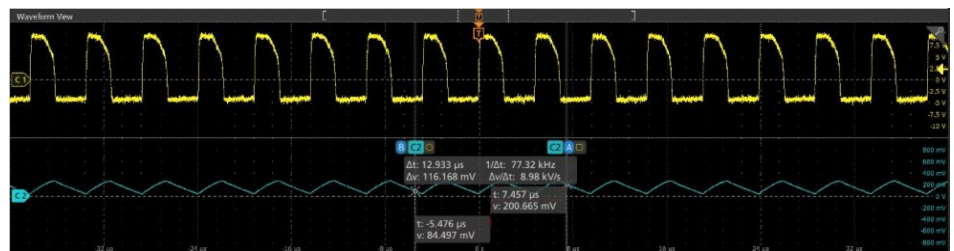
1. Select the plot to which you want to add cursors.
2. Tap the **Add New... Cursors** button, or push the front-panel **Cursors** button.

The cursors are displayed on the plot. By default, the Cursor read-out box is displayed on the left corner of the plot view.



3. Use Multipurpose Knobs **A** and **B** to move the cursors on the waveform, or touch and drag a cursor.

**NOTE.** Moving the waveform cursor in the Waveform View moves the respective cursors in the plot view.



Cursor readouts show the position with B and H coordinates, the difference of B and H between the cursors, and the permeability value.

4. To further configure cursors, double-tap on either cursor line or the cursor readouts to open the **Cursors** configuration menu.
5. Tap the Help icon on the menu title for more information on the menu settings.
6. To stop showing cursors, push the front panel **Cursor** button, or press and hold on a cursor to open the right-click menu and turn cursors off, or open the Cursors configuration menu and set Display to **Off**.

**SOA plot configuration menu (optional)**

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
<b>AutoScale</b>	Toggles <b>AutoScaleOn</b> or <b>Off</b> . Turn <b>AutoScale</b> off to manually set the X and Y-axis range to view an area of interest.  When <b>AutoScale</b> 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.
<b>Gridlines</b>	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
<b>Display</b>	Vectors draws waveforms with lines between record points.  Dots draws waveform record points as dots on the screen.
<b>X-Axis (Voltage) Scale</b>	To change the scale from linear to log or vice versa, click on the button (linear or log).  This scale applies on the plot data as well as on the mask data.
<b>Y-Axis (Current) Scale</b>	To change the scale from linear to log or vice versa, click on the button (linear or log).  This scale 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
<b>Save Plot Image</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot image.  Enter a file name in the <b>File Name</b> field. Select the image file format (PNG, BMP, or JPG).  Tap OK to save the plot image.
<b>Save Plot Data</b>	Tap to open the <b>Save As</b> menu. Navigate to the location where you want to save the plot data.  Enter a file name in the <b>File Name</b> 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) (optional)**

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
<b>AutoScale</b>	<p>Toggles <b>AutoScaleOn</b> or <b>Off</b>. Turn <b>AutoScale</b> off to manually set the X and Y-axis range to view an area of interest.</p> <p>When <b>AutoScale</b> 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.</p>
<b>Gridlines</b>	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
<b>Display</b>	<p>Vectors draws waveforms with lines between record points.</p> <p>Dots draws waveform record points as dots on the screen.</p>
<b>X-Axis (Voltage) From, To</b>	<p>Sets the beginning and end scale range to display in the plot for the X-Axis scales.</p> <p>It is displayed only when AutoScale is off (unchecked).</p>
<b>Y-Axis (Voltage) From, To</b>	<p>Sets the beginning and end scale range to display in the plot for the Y-Axis scales.</p> <p>It is displayed only when AutoScale is off (unchecked).</p>

**Save panel fields and controls.** Use the Save panel controls to save the plot image or data to a file, for inclusion in reports or further analysis in other applications.

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

**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
<b>Save Location</b>	<p>Lists the location where the file will be saved. The default value is the last location to which a file was saved.</p> <p>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.</p> <p>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.</p>
<b>Browse</b>	<p>Opens the <a href="#">Browse Save As Location</a> configuration menu to navigate to and select a location at which to save the file.</p>
<b>File name</b>	<p>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.</p> <p>Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names.</p> <p>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.</p>
<b>Save As Type</b>	<p>Lists the available graphic formats to which you can save files.</p> <p>Tap the field and select the graphic save format.</p>
<b>Cancel</b>	<p>Cancels the file save action and closes the configuration menu.</p>
<b>OK</b>	<p>Saves the file to the specified location, closes the Save As menu, and displays a confirmation message.</p>

**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
<b>Save Location</b>	<p>Lists the location where the file will be saved. The default value is the last location to which a file was saved.</p> <p>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.</p> <p>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.</p>
<b>Browse</b>	<p>Opens the <a href="#">Browse Save As Location</a> configuration menu to navigate to and select a location at which to save the file.</p>
<b>File name</b>	<p>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.</p> <p>Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names.</p> <p>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.</p>
<b>Cancel</b>	<p>Cancels the file save action and closes the configuration menu.</p>
<b>OK</b>	<p>Saves the file to the specified location, closes the Save As menu, and displays a confirmation message.</p>

## 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
<b>Display</b>	Turns On or Off displaying the waveform.
<b>Vertical Scale</b>	Set the vertical scale by using the assigned multipurpose knob, a virtual keypad, or tap the up or down arrows.
<b>Label</b>	<p>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.</p> <p>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.</p>
<b>Units</b>	Set the units label that you want to display on the vertical scale.
<b>Position</b>	Set the vertical position of the waveform using the assigned multipurpose knob or the virtual keypad.
<b>Set to 0</b>	Tap to set the vertical position to 0 (vertical center of the graticule).
<b>Reference File</b>	<p>Shows the path and file name of the current Reference waveform.</p> <p>Double-tap the field to open the on-screen keyboard to enter or edit the path to open a different waveform file.</p> <p>Tap the down arrow icon to list the 20 most recently accessed reference waveform files.</p>
<b>Browse</b>	Opens the Browse Waveform File dialog. Use this dialog to navigate to and select a waveform file to load. See <a href="#">Recall configuration menu (Ref waveform configuration menu)</a> on page 475.
<b>Sample Rate, Record Length</b>	<p>Readout-only text that show the sample rate and record length values of the reference waveform.</p> <p>These fields are replaced with <b>Center Frequency</b>, <b>Span</b>, and <b>RBW</b> readouts for spectrum reference waveforms.</p>
<b>Deskew</b>	<p>Changes the horizontal position of the reference waveform. Use this function to align the reference waveform to meet your measurement needs.</p> <p>This field is not displayed for spectrum reference waveforms.</p>

Field or control	Description
<b>Set to 0</b>	<p>Sets the reference waveform deskew value to zero.</p> <p>This field is not displayed for spectrum reference waveforms.</p>
<b>Selected Frame</b>	<p>Sets the frame to view of a recalled FastFrame waveform file. This control is only available if the recalled waveform includes FastFrame acquisitions.</p> <hr/> <p><b>NOTE.</b> <i>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.</i></p> <hr/> <p>Navigating through the frames of a reference waveform does not require the FastFrame feature to be active.</p> <hr/> <p>If more than one fast frame reference waveform is active, the Selected Frame in each waveform's configuration menu is independent of the others.</p>

**Recall configuration menu  
(Ref waveform  
configuration menu)**

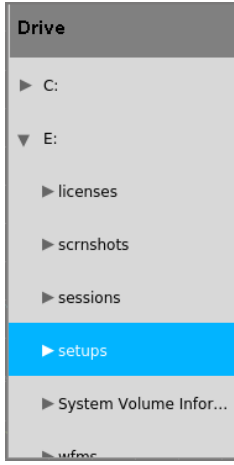


Use this menu to locate and load a reference waveform file.

Prerequisite: a Ref badge must be present on the Settings bar. See [Add a math, reference, or bus waveform](#) on page 111.

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
<b>Look in</b>	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, including any network-connected drives. Use to quickly navigate to a location.  Tap to list the contents of the directory in the file list area.  Double-tap an item, or tap the small arrow to the left of the directory, to display the subdirectories under it. Double-tap again to close that directory structure.  If there is a scroll bar, drag the list up and down to show more entries.
	Use the arrow buttons to navigate the directory structure.  The left arrow navigates back to the previously visited folder.  The Right arrow navigates forward to the previously visited folder.  The Up arrow navigates up one level from the current folder.
	Use to create a new directory (folder) at the current location.  Opens the new directory after it is created.
<b>File Name</b>	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.
<b>Files of Type</b>	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.
<b>Cancel</b>	Cancels any changes, closes the menu, and returns to the prior menu without loading a file.

Field or control	Description
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 320

[Edge Search configuration menu](#) on page 355

[Logic search configuration menu](#) on page 356

[Pulse Width Search configuration menu](#) on page 359

[Rise/Fall Time Search configuration menu](#) on page 361

[Runt Search configuration menu](#) on page 363

[Setup and Hold Search configuration menu](#) on page 365

[Timeout Search configuration menu](#) on page 368

[Window Search configuration menu](#) on page 369

[DDR Read Search configuration menu \(Requires option SUP6-DBDDR3\)](#) on page 372

[DDR Write Search configuration menu \(Requires option SUP6-DBDDR3\)](#) on page 374

[DDR Read & Write Search configuration menu \(Requires option SUP6-DBDDR3\)](#) on page 376

## Spectrum View menus

Use the Spectrum View menus to activate and configure frequency-domain spectrum traces for analog channel time-domain signals.

### Spectrum View window configuration menu

Use the Spectrum View window configuration menu to change waveform intensity, graticule style and intensity, and peak marker settings that apply to all spectrum waveforms in the Spectrum View.

To open the Spectrum View window configuration menu, double-tap anywhere on the **Spectrum View** window other than on a waveform readout, or trace handle.

#### View Settings panel, fields and controls.

Field or control	Description
<b>Display Mode</b>	Sets the spectrum View window mode to <b>Overlay</b> (all waveforms overlaid on a single graticule) or <b>Stacked</b> (all waveforms in their own row or slice of the Spectrum View window).
<b>Waveform Intensity</b>	Sets the brightness level of all waveforms.
<b>Graticule Style</b>	<p>Sets the graticule style to display.</p> <p><b>Grid</b> displays a grid, cross hairs, and frame.</p> <p><b>Time</b> displays a vertical grid of time marks, cross hairs, and a frame.</p> <p><b>Full</b> displays a frame and a grid. This style is useful for making quick, full-screen measurements with cursors and automatic readouts when cross hairs are not needed.</p> <p><b>None</b> displays a frame without a grid and cross hairs.</p> <p>When you display a grid in the <b>Spectrum View</b>, the horizontal zero line is the Reference level, and the vertical zero line is the center frequency.</p>
<b>Graticule Intensity</b>	<p>Sets the brightness of all graticules.</p> <p>Changing the graticule intensity also changes the intensity of vertical and horizontal scales and readouts. Graticule intensity does not change the intensity of note text on the screen.</p>



## Markers panel, fields and controls.

Field or control	Description
<b>Peak Markers</b>	<p>Enables (<b>On</b>) or disables (<b>Off</b>) showing triangular marker icons on the top edge of each spectrum trace for a specified quantity of waveform peaks. Default is <b>On</b>.</p> <p>The highest peak on the currently selected trace is the reference marker, and is marked with a red triangle.</p> <p>Other peaks are shown in the color of the selected spectrum trace.</p>
<b>Quantity</b>	<p>Sets the maximum number of highest peak markers to show for each trace in the Spectrum View window. The default value is 5.</p> <p>When the display mode is <b>Stacked</b>, this control sets the maximum number of peaks that can be marked in each waveform slice.</p> <p>When the display mode is <b>Overlay</b>, this control set the maximum number of peaks that can be marked on the currently selected spectrum waveform trace.</p> <p>If there are fewer peaks in the Spectrum View window than this control is set for, only the displayed peaks will be marked.</p>
<b>Threshold</b>	<p>Sets the minimum amplitude that a signal must exceed to be a valid peak. The default value is -50.0 dBm.</p> <p>The instrument shows a temporary horizontal line on the spectrum traces (like the Trigger Level indicator) when changing this value, to show the peak threshold level in relationship to the spectrum trace.</p>
<b>Excursion</b>	<p>Sets the minimum magnitude (<math> PK_{MAX}  -  Peak_{MIN} </math>) a peak must exceed to be marked. The default value is 30.0 dB.</p> <p><b>Threshold</b> and <b>Excursion</b> are AND functions; both must be true for a marker to be placed on a peak (including the Reference marker).</p>
<b>Readout</b>	<p>Sets the marker readouts to show <b>Absolute</b> or <b>Delta</b> values. The default value is <b>Absolute</b>.</p> <p><b>Absolute</b> readouts show the actual frequency and peak value.  <b>Delta</b> readouts show the frequency and amplitude values of the markers relative to the Reference Marker values.</p>

**Selected Trace configuration menu (Spectrum View)**

Use this menu to display any combination of normal, average, minimum, and maximum spectrum trace data to show.

To open the **Spectrum View Selected** Trace configuration menu, double-tap the spectrum trace handle.

**Selected Trace menu, fields and controls.**

Field or control	Description
<b>Normal</b>	Sets the spectrum trace of the selected channel to show the first sample point data in each sample interval.
<b>Max Hold</b>	Sets the spectrum trace of the selected channel to show the highest amplitude points in each sample interval over several acquisitions.
<b>Min Hold</b>	Sets the spectrum trace of the selected channel to show the lowest amplitude points in each sample interval over several acquisitions. Sets the trace mode to show all available sample point data in the spectrum trace.
<b>Average</b>	Sets the spectrum trace of the selected channel to show the average of the normal points in each sample interval over several acquisitions.  The number of averages is set in the Spectrum View panel of the Channel configuration menu. See <a href="#">Spectrum View panel (Channel configuration menu)</a> on page 383.

**Spectrum View Cursors menu**

Use the Spectrum View cursors to take manual measurements on spectrum signals.

To open the **Spectrum View Cursors** menu, double-tap on a cursor or its readout in the **Spectrum View** window.

**Cursors menu (Spectrum View), fields and controls.**

Field or control	Description
<b>Display</b>	Tap to toggle the spectrum cursor display <b>On</b> or <b>Off</b> .
<b>Source Waveform</b>	<p>Sets the cursor trace mode (the trace(s) on which to display the cursors).</p> <p><b>Same</b> places both cursors on the same trace.</p> <p><b>Split</b> allows each cursor to be on a different trace.</p>
<b>Source</b>	<p>Moves the spectrum trace cursors and readouts to the selected source (default) or a specific selected from the list.</p> <p><b>Selected Waveform</b> lets you tap the screen or use a mouse to quickly move the cursors to a selected trace (<b>Overlay</b> mode) or slice (<b>Stacked</b> mode).</p> <p>Only available when <b>Source Waveform = Same</b>.</p>
<b>Cursor A Source, Cursor B Source</b>	<p>Sets the waveform sources for Cursor A and B when in split cursor mode. Only available when <b>Source Waveform = Split</b>.</p>
<b>Cursor A X-Position</b>	Sets a specific x-axis (frequency) position for Cursor A using the multipurpose knob or the virtual keypad.
<b>Cursor B X-Position</b>	Sets a specific x-axis (frequency) position for Cursor B using the multipurpose knob or the virtual keypad.
<b>Readout</b>	<p>Sets the cursor readouts to show <b>Absolute</b> or <b>Delta</b> values. The default value is <b>Absolute</b>.</p> <p><b>Delta</b> readouts are relative to the Reference Marker.</p>

### Spectrum cursors guidelines.

- **Spectrum View** only supports waveform cursors.
- Cursor readouts only appear in the selected spectrum slice. The cursor bar extends through all slices (similar to waveform cursors in **Stacked Waveform View** slices).
- Cursors maintain their screen position if you change the center frequency or span.
- Cursor measurements are taken on the currently selected trace (**Normal**, **Average**, **Max Hold** or **Min Hold**) which is indicated by the MANm trace handle. See *Spectrum trace handle and trace types* on page 172.
- Cursor A becomes the Reference marker. The cursor A readout shows Frequency, Amplitude and Noise Density values. The automatic peak markers still show the same number of peaks as before, but they use the cursor A reference marker for delta measurements.
- The cursor B readout depends on the Absolute/Delta mode:
  - When cursor readouts are set to **Absolute** mode, the cursor B readout shows Frequency, Amplitude, and Noise Density values.
  - When cursor readouts are set to **Delta** mode, the cursor B readout shows the delta Frequency from cursor A, the delta Amplitude from cursor A, and the Phase Noise of cursor B relative to cursor A (dBc/Hz).
  - Split cursors readouts only display frequency readouts when the cursor sources are using different vertical units.

### Spectrum badge menu

Use the Spectrum badge menu to set spectrum parameters including source, center frequency, span, RBW, FFT window type, and spectrum time.

To open the Spectrum badge menu, double-tap the badge.

## Spectrum badge menu, fields and controls.

Field or control	Description
<b>Lock Center Frequency Across All Channels</b>	<p>Enables or disables locking all spectrum center frequencies to the currently selected channel's center frequency. The default value is enabled (selected).</p> <p>When selected, sets all spectrum traces to use the same center frequency as set in the <b>Center Frequency</b> field.</p> <p>When not selected, each spectrum trace can be set to a different center frequency.</p>
<b>Source</b>	<p>Sets the channel source to use for settings in the menu. Select a source from the drop-down list.</p> <hr/> <p><b>NOTE.</b> The <i>Source</i> field value is the last source channel set in the menu. The <i>Source</i> field does not update to reflect the active Spectrum trace channel when the menu is opened.</p> <hr/> <p>Only available when <b>Lock Center Frequency Across All Channels</b> is disabled (not selected).</p>
<b>Center Frequency</b>	<p>Sets or shows the center frequency value of the selected trace.</p> <p>If <b>Lock Center Frequency Across All Channels</b> is unchecked, then this field updates as different channels are selected in the <b>Source</b> field.</p> <p>If <b>Lock Center Frequency Across All Channels</b> is checked, then this field sets all spectrum traces to use the frequency shown in the field.</p> <p>Use the field to enter a specific center frequency value for the current source or all channels, depending on the setting of the <b>Lock Center Frequency Across All Channels</b> control.</p>
<b>Span</b>	<p>Sets the range of frequencies to display, centered on the center frequency value. Use the arrow controls to increment or decrement the span value in a 1-2-5 sequence, or double-tap the field to enter a value.</p> <hr/> <p><b>NOTE.</b> The <i>Span</i> setting applies to all spectrum traces. Making a span change for one source changes the span for all signal sources.</p>

Field or control	Description
<b>Window</b>	<p>Sets the FFT window to apply to the signal. Windows are mathematical filters applied to provide more accurate frequency or magnitude traces. Each window offers a trade-off between frequency resolution and magnitude accuracy.</p> <p>The choice of which window to use depends upon what you want to measure and your source's signal characteristics. See <a href="#">FFT windows</a> on page 453 for more FFT window information.</p> <hr/> <p><b>NOTE.</b> The <b>Window</b> setting applies to all spectrum traces. Changing the FFT for one source changes the FFT for all signal sources.</p>
<b>RBW Mode</b>	<p>Sets either <b>Auto</b> or <b>Manual</b> resolution bandwidth mode.</p> <p><b>Auto</b> sets the resolution bandwidth to change automatically as you change the span. The default behavior is <math>RBW = \text{Span}/1000</math>.</p> <p><b>Manual</b> lets you enter a specific resolution bandwidth.</p>
<b>Span:(RBW)</b>	<p>Sets the resolution bandwidth as a ratio of the span. Use the arrow controls to increment or decrement the span ratio in a 1-2-5 sequence, or double-tap the field to enter a value.</p> <p>Only available when <b>RBW Mode</b> is set to <b>Auto</b>.</p>
<b>RBW</b>	<p>Sets a specific resolution bandwidth value. Use the arrow controls to increment or decrement the RBW in a 1-2-5 sequence, or double-tap the field to enter a value.</p> <p>Only available when <b>RBW Mode</b> is set to <b>Manual</b>.</p>
<b>Lock Spectrum Time Across All Channels</b>	<p>Enables or disables locking all spectrum times to the currently selected channel's spectrum time location, shown as a percent of the record length. The default value is enabled (selected).</p> <p>When enabled, this function sets all spectrum traces to use the spectrum time as set in the <b>Spectrum Time</b> field.</p> <p>When not selected, each spectrum trace can be set to a different spectrum time. See <a href="#">Spectrum View and Spectrum Time</a> on page 169.</p>
<b>Source</b> (Lock Spectrum Time)	<p>Sets the source to use for the spectrum time value. Select a source from the drop-down list.</p> <p>Only available when <b>Lock Spectrum Time Across All Channels</b> is enabled (selected).</p>

Field or control	Description
<b>Spectrum Time</b>	<p>Sets the channel's spectrum time location, shown as a percent of the record length. Double-tap the field to enter a value.</p> <p>If <b>Lock Spectrum Time Across All Channels</b> is unchecked, then this field updates as different channels are selected in the upper-menu <b>Source</b> field.</p> <p>If <b>Lock Spectrum Time Across All Channels</b> is checked, then this field sets all spectrum traces to use the spectrum time shown in the upper-menu <b>Source</b> field.</p>

## 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](#)

### Edge Trigger configuration menu

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

To open the Edge trigger menu:

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

**Settings panel (Edge Trigger configuration menu) fields and controls.**

Field or control	Description
<b>Trigger Type</b>	Set to <b>Edge</b> .
<b>Source</b>	<p>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.</p> <p><b>AC Line source (Edge trigger):</b> The instrument uses the AC line to generate a trigger based on the power line frequency. The trigger level is fixed at zero volts. This source is often used to look at signals related to the power line frequency (for example, signals from devices such as lighting equipment and power supplies). Because the instrument generates the trigger from the power line, you do not have to use a channel input.</p> <p><b>AUX In:</b> The MSO58LP and 6 Series MSO also have an <b>AUX In</b> (auxiliary) edge trigger source. <b>AUX In</b> provides an extra trigger signal source when you need to use the input channels for other signals. For example, you might want to trigger on a clock while displaying related logic signals. The <b>AUX In</b> input connector is not compatible with most probes, and you cannot display the auxiliary trigger signal in the Waveform view.</p>
<b>Coupling</b>	<p>Set the conditioning to apply to the source signal trigger circuit from the source signal.</p> <p><b>DC</b> coupling passes all input signals directly to the trigger circuitry.</p> <p><b>HF Reject</b> coupling attenuates signals above 50 kHz before passing the signal to the trigger circuitry.</p> <p><b>LF Reject</b> coupling attenuates signals below 50 kHz before passing the signal to the trigger circuitry.</p> <p><b>Noise Reject</b> coupling provides stable triggering by increasing the trigger hysteresis. Increased hysteresis reduces the trigger sensitivity to noise so may require greater signal amplitude.</p>
<b>Level</b>	Sets the amplitude level that the signal must pass through to be considered a valid transition.
<b>Set to 50%</b>	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
<b>Slope</b>	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
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**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](#)

**Bus Trigger configuration**

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

---

**NOTE.** *You must add a bus to the Waveform view before you can trigger on it. [Add a math, reference, or bus waveform](#) on page 111.*

---

To open the Bus trigger menu:

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

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

[ARINC 429 serial bus trigger settings panel](#) on page 490

[Audio serial bus trigger settings panel](#) on page 493

[CAN serial bus trigger settings panel](#) on page 495

[Ethernet serial bus trigger settings panel](#) on page 498

[FlexRay serial bus trigger settings panel](#) on page 502

[I2C serial bus trigger settings panel](#) on page 505

[LIN serial bus trigger settings panel](#) on page 507

[MIL-STD-1553 serial bus trigger settings panel](#) on page 509

[Parallel serial bus trigger settings panel](#) on page 512

[RS-232 serial bus trigger settings panel](#) on page 514

[SENT serial bus trigger settings panel](#) on page 516

[SPI serial bus trigger settings panel](#) on page 519

[SPMI serial bus trigger settings panel](#) on page 521

[USB serial bus trigger settings panel](#) on page 524

**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*

**ARINC 429 serial bus trigger settings panel.** Requires option SRAERO. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>ARINC429</b> bus on which you want to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Trigger When Label</b>	Trigger when the label data meets the specified condition  Available when <b>Trigger On = Label</b> .
<b>Label</b>	Sets the label pattern on which to trigger.  Tap the <b>Binary, Hex, or Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Label or Label &amp; Data</b> .
<b>Label Low</b>	Sets the low value of the label pattern range on which to trigger.  Tap the <b>Binary, Hex, or Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Label</b> and <b>Trigger When Label = Inside Range or Outside Range</b> .
<b>Label High</b>	Sets the high value of the label pattern range on which to trigger.  Tap the <b>Binary, Hex, or Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Label</b> and <b>Trigger When Label = Inside Range or Outside Range</b> .
<b>Data</b>	Sets to trigger when the specified data bits condition occurs.  Tap the <b>Binary or Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Data or Label &amp; Data</b> .
<b>Data Low</b>	Sets the low value of the data pattern range on which to trigger.  Tap the <b>Binary, Hex, or Octal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Data</b> and <b>Trigger When Data = Inside Range or Outside Range</b> .

Field or control	Description
<b>Data High</b>	<p>Sets the high value of the data pattern range on which to trigger.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Octal</b> field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Data</b> and <b>Trigger When Data = Inside Range</b> or <b>Outside Range</b>.</p>
<b>SSM</b>	<p>Sets to trigger when the specified Sign/Status Matrix (SSM) bit condition occurs.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Data</b>.</p>
<b>SDI</b>	<p>Sets to trigger when the specified Source/Destination Identifier (SDI) bit condition occurs.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Data</b>.</p>
<b>Error Type</b>	<p>Sets the error condition on which to trigger.</p> <p>Available when <b>Trigger On = Error</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change in data or bit fields.</p> <p>Use the <b>B</b> knob to change the value of the digit(s) in the selected field.</p>

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**Audio serial bus trigger settings panel.** Requires option SRAUDIO. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>Audio</b> bus on which to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Data</b>	<p>Sets the data pattern on which to trigger. Use in conjunction with the <b>Trigger When</b> field to specify the exact trigger condition.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Data</b>.</p>
<b>Word</b>	<p>Sets the audio word channel on which to trigger (<b>Either</b>, <b>Left</b>, <b>Right</b>).</p> <p>Available when <b>Trigger On = Data</b>.</p>
<b>Trigger When</b>	<p>Sets the trigger when condition for the specified data pattern.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a high and low boundary pattern for the specified trigger type.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Data</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>



**CAN serial bus trigger settings panel.** Requires option SRAUTO. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>CAN</b> bus on which you want to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Frame Type</b>	Sets the frame type on which to trigger.  Available when <b>Trigger On = Type of Frame</b> .
<b>Direction</b>	Sets the transfer direction on which to trigger (Read, Write, Either).  Available when <b>Trigger On = Identifier</b> .
<b>Identifier Format</b>	Sets the identifier for standard (11-bit) or extended (29-bit for CAN 2.0B) length.  Available when <b>Trigger On = Identifier</b> or <b>ID &amp; Data</b> .
<b>Identifier</b>	Sets the identifier pattern on which to trigger. The number of bits shown depends on the <b>Identifier Format</b> setting.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Identifier</b> or <b>ID &amp; Data</b> .
<b>Data Bytes</b>	Sets the number of data bytes on which to trigger (one to eight bytes). Use the A knob to change the value.  Available when <b>Trigger On = Data</b> or <b>ID &amp; Data</b> .
<b>Data</b>	Sets the data pattern on which to trigger. The number of bits shown depends on the <b>Data Bytes</b> setting.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Data</b> or <b>ID &amp; Data</b> .
<b>Data Offset</b>	Sets the byte offset value.  Available when <b>Trigger On = Data</b> or <b>ID &amp; Data</b> .
<b>Trigger When</b>	Sets the trigger when condition.  Available when <b>Trigger On = Data</b> or <b>ID &amp; Data</b> .
<b>Bit Rate Switch (BRS) Bit</b>	Sets the BRS bit state on which to trigger.  Available when <b>Source</b> is a <b>CAN FD</b> bus and <b>Trigger On = FD Bits</b> .
<b>Error Status Indicator (ESI) Bit</b>	Sets the ESI bit state on which to trigger.  Available when <b>Source</b> is a <b>CAN FD</b> bus and <b>Trigger On = FD Bits</b> .

Field or control	Description
<b>Error Type</b>	Sets the error type on which to trigger.  Available when <b>Trigger On = Error</b> .
<b>A, B knob controls</b>	Use the <b>A</b> knob to select (highlight) the digit(s) to change.  Use the <b>B</b> knob to change the value of the digit(s).

Field or control	Description
<b>Trigger Mode</b>	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:  <b>Auto</b> 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.  <b>Normal</b> 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.
<b>Force Trigger</b>	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
<b>Holdoff</b>	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.  <b>Random</b> 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.  <b>Time</b> 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.
<b>Holdoff Time</b>	Tap the <b>Holdoff Time</b> 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.

---

Field or control	Description
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**Ethernet serial bus trigger settings panel.** Requires option SRENET. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>Ethernet</b> bus on which to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>MAC Address Destination, Source</b>	<p>Sets the MAC destination and/or source address pattern on which to trigger.</p> <p>Tap a <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = MAC Addresses</b>.</p>
<b>MAC Length/Type</b>	<p>Sets the MAC length/type pattern on which to trigger.</p> <p>Tap a <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = MAC Length / Type</b>.</p>
<b>MAC Length/Type Low, MAC Length/Type High</b>	<p>Sets the boundary MAC length type conditions when testing for in-range or out of range conditions.</p> <p>Available when <b>Trigger On = MAC Length / Type</b> and <b>Trigger When = Inside Range</b> or <b>Outside Range</b>.</p>
<b>IP Protocol</b>	<p>Sets the IP protocol pattern on which to trigger.</p> <p>Tap a <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = IP Header</b>.</p>
<b>Source, Destination Address</b>	<p>Sets the source and/or destination IP address pattern on which to trigger.</p> <p>Tap a <b>Hex</b> or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = IP Header</b>.</p>
<b>Source Port, Destination Port</b>	<p>Sets the source and/or destination TCP header port pattern on which to trigger.</p> <p>Tap a <b>Binary</b>, <b>Hex</b> or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = TCP Header</b>.</p>

Field or control	Description
<b>Sequence Number</b>	<p>Sets the TCP header sequence number pattern on which to trigger.</p> <p>Tap a <b>Binary</b>, <b>Hex</b> or <b>Decimal</b> field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = TCP Header</b>.</p>
<b>Ack Number</b>	<p>Sets the TCP header ack number pattern on which to trigger.</p> <p>Tap a <b>Binary</b>, <b>Hex</b> or <b>Decimal</b> field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = TCP Header</b>.</p>
<b>Data Bytes</b>	<p>Sets the number of client data bytes on which to trigger (one to sixteen bytes).</p> <p>Tap the field and use the A knob to change the value. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Client Data</b>.</p>
<b>Byte Offset</b>	<p>Sets the client data byte offset (Don't Care or the number of bytes).</p> <p>Tap the field and use the A knob to change the value. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Client Data</b>.</p>
<b>Client Data</b>	<p>Sets the data pattern on which to trigger. The number of bits shown depends on the <b>Data Bytes</b> setting. Use in conjunction with the <b>Trigger When</b> field to specify the exact trigger condition.</p> <p>Tap the <b>Binary</b>, <b>Hex</b> or <b>ASCII</b> field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Client Data</b>.</p>
<b>Client Data Low, Client Data High</b>	<p>Sets the boundary data values when testing for in-range or out of range conditions. The number of bits shown depends on the <b>Data Bytes</b> setting. Use in conjunction with the <b>Trigger When</b> field to specify the exact trigger condition.</p> <p>Tap the <b>Binary</b>, <b>Hex</b> or <b>ASCII</b> field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Client Data</b> and <b>Trigger When = Inside Range</b> or <b>Outside Range</b>.</p>

Field or control	Description
<b>Trigger When</b>	<p>Sets the trigger when condition.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a high and low boundary pattern for the specified trigger type.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = MAC Length/Type</b> or <b>Client Data</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>

Field or control	Description
<b>Holdoff Time</b>	Tap the <b>Holdoff Time</b> 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.
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**FlexRay serial bus trigger settings panel.** Requires option SRAUTO. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>FlexRay</b> bus on which to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Indicator Bits</b>	<p>Select the indicator bits type on which to trigger.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Indicator Bits</b> or <b>Header Fields</b>.</p>
<b>Identifier</b>	<p>Sets the frame identifier pattern on which to trigger.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Frame ID</b>, <b>Header Fields</b>, or <b>Identifier &amp; Data</b>.</p>
<b>Cycle Count</b>	<p>Sets the cycle count pattern on which to trigger. Use in conjunction with the Trigger When field to specify the exact trigger condition.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Cycle Count</b> or <b>Header Fields</b>.</p>
<b>Payload Length</b>	<p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Header Fields</b>.</p>
<b>Header CRC</b>	<p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Header Fields</b>.</p>
<b>Data Bytes</b>	<p>Sets the number of data bytes on which to trigger (one to sixteen bytes). Use the <b>A</b> knob to change the value. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Data</b> or <b>Identifier &amp; Data</b>.</p>



Field or control	Description
<b>Data</b>	<p>Sets the data pattern on which to trigger. The number of bits shown depends on the <b>Data Bytes</b> setting. Use in conjunction with the <b>Trigger When</b> field to specify the exact trigger condition.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>Byte Offset</b>	<p>Sets the byte offset (<b>Don't Care</b> or the number of bytes). Tap the input box and use the <b>A</b> knob to change the value.</p> <p>Available when <b>Trigger On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>Trigger When</b>	<p>Sets the trigger when condition.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a high and low boundary pattern for the specified trigger type.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Frame ID, Cycle Count, Data, or Identifier &amp; Data</b>.</p>
<b>Frame Type</b>	<p>Sets the end of frame type on which to trigger (Static, Dynamic (DTS), ALL).</p> <p>Available when <b>Trigger On = End of Frame</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**I<sup>2</sup>C serial bus trigger settings panel.** Requires option SREMBD. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>I<sup>2</sup>C</b> bus on which to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Direction</b>	Sets the transfer direction on which to trigger (Read, Write, Either).  Available when <b>Trigger On = Address</b> .
<b>Addressing Mode</b>	Sets the slave device address length (7 bits or 10 bits long).  Available when <b>Trigger On = Address</b> or <b>Address &amp; Data</b> .
<b>Address</b>	Sets the address pattern on which to trigger. The number of bits shown depends on the <b>Address Mode</b> setting.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Address</b> or <b>Address &amp; Data</b> .
<b>Data Bytes</b>	Sets the number of data bytes on which to trigger (one to five bytes). Use the <b>A</b> knob to change the value.  Available when <b>Trigger On = Data</b> or <b>Address &amp; Data</b> .
<b>Data</b>	Sets the data pattern on which to trigger. The number of bits shown depends on the <b>Data Bytes</b> setting.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Data</b> or <b>Address &amp; Data</b> .
<b>A, B knob controls</b>	Use the <b>A</b> knob to select (highlight) the digit(s) to change.  Use the <b>B</b> knob to change the value of the digit(s). Or double-tap on the field and use the virtual keypad to enter values.

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**LIN serial bus trigger settings panel.** Requires option SRAUTO. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>LIN</b> bus on which to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Identifier</b>	<p>Sets the identifier pattern on which to trigger.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Identifier</b> or <b>Identifier &amp; Data</b>.</p>
<b>Data</b>	<p>Sets the data pattern on which to trigger. The number of bits shown depends on the <b>Data Bytes</b> setting.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>Trigger When</b>	<p>Sets the trigger when condition.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a high and low boundary pattern for the specified trigger type.</p> <p>Available when <b>Trigger On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>Data Bytes</b>	<p>Sets the number of data bytes on which to trigger (one to eight bytes). Use the <b>A</b> knob to change the value.</p> <p>Available when <b>Trigger On = Data</b> or <b>Identifier &amp; Data</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s). Or double-tap on the field and use the virtual keypad to enter values.</p>
<b>Error Type</b>	<p>Sets the LIN error type on which to trigger.</p> <p>Available when <b>Trigger On = Error</b>.</p>

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**MIL-STD-1553 serial bus trigger settings panel.** Requires option SRAERO. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>MIL-STD-1553</b> bus on which you want to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Sync</b>	Sets to trigger on a Sync condition.  Available when <b>Trigger On = Sync</b> .
<b>Transmit/Receive Bit</b>	Sets the transmit or receive bit on which to trigger.  Available when <b>Trigger On = Command</b> .
<b>Trigger When RT Address</b>	Sets to trigger when the specified RT address condition occurs.  Available when <b>Trigger On = Command</b> or <b>Status</b> .
<b>Parity</b>	Sets to trigger on the selected parity bit logic state.  Available when <b>Trigger On = Command, Status, or Data</b> .
<b>Address</b>	Sets the address pattern on which to trigger.  Tap the <b>Binary, Hex, or Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Command</b> or <b>Status</b> .
<b>Low Address</b>	Sets the low value of the address pattern range on which to trigger.  Tap the <b>Binary, Hex, or Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger When RT Address = Inside Range</b> or <b>Outside Range</b> .
<b>High Address</b>	Sets the high address of the address pattern range on which to trigger.  Tap the <b>Binary, Hex, or Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger When RT Address = Inside Range</b> or <b>Outside Range</b> .
<b>Subaddress/Mode</b>	Sets the subaddress or mode pattern on which to trigger.  Tap the <b>Binary, Hex, or Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Command</b> .

Field or control	Description
<b>Status Word Bits</b>	<p>Sets the status word pattern on which to trigger.</p> <p>Tap the <b>Binary</b>, <b>Hex</b>, or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Selecting a bit shows a short description of that bit's function. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Status</b>.</p>
<b>Data</b>	<p>Sets the data pattern on which to trigger.</p> <p>Available when <b>Trigger On = Data</b>.</p>
<b>Trigger When</b>	<p>Sets to trigger when the specified RT/IMG signal time condition occurs.</p> <p>Available when <b>Trigger On = Time (RT/IMG)</b>.</p>
<b>Minimum Time</b>	<p>Sets the minimum time for a valid RT/IMG signal.</p> <p>Available when <b>Trigger On = Time (RT/IMG)</b>.</p>
<b>Maximum Time</b>	<p>Sets the maximum time for a valid RT/IMG signal.</p> <p>Available when <b>Trigger On = Time (RT/IMG)</b>.</p>
<b>Error Type</b>	<p>Sets the error condition on which to trigger.</p> <p>Available when <b>Trigger On = Error</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change in data or bit fields.</p> <p>Use the <b>B</b> knob to change the value of the digit(s). Or double-tap on the field and use the virtual keypad to enter values.</p>



Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	Tap the <b>Holdoff Time</b> 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.
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**Parallel serial bus trigger settings panel.** Parallel bus triggering is standard on all instruments. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the type of information on which to trigger.
<b>Data</b>	<p>Sets the data pattern on which to trigger. The number of bits shown depends on how the parallel bus is defined.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>

Field or control	Description
<b>Holdoff Time</b>	Tap the <b>Holdoff Time</b> 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.
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**RS-232 serial bus trigger settings panel.** Requires option SRCOMP. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

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

Field or control	Description
<b>Trigger Mode</b>	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:  <b>Auto</b> 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.  <b>Normal</b> 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.
<b>Force Trigger</b>	Forces a trigger event regardless of whether the waveform meets any trigger conditions.

Field or control	Description
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**SENT serial bus trigger settings panel.** Requires option SRAUTOSEN. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>SENT</b> bus on which to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Channel</b>	Sets the SENT channel type on which to trigger.  Available when <b>Trigger On = Start of Packet</b> .
<b>Status / Communications</b>	Sets the value of the status/communications nibble on which to trigger.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Fast Channel</b> .
<b>Fast Channel 1</b>	Sets the condition and value of the fast channel 1 data on which to trigger.  Tap the down arrow and select the condition on which to trigger (=, ≠, >, <, ≥, ≤). The default is =. Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Fast Channel</b> .
<b>Fast Channel 2</b>	Sets the condition and value of the fast channel 2 data on which to trigger.  Tap the down arrow and select the condition on which to trigger (=, ≠, >, <, ≥, ≤). The default is =. Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Fast Channel</b> .
<b>Counter</b>	Sets the condition and value of the counter data on which to trigger.  Tap the down arrow and select the condition on which to trigger (=, ≠, >, <, ≥, ≤). The default is =. Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Fast Channel</b> .

Field or control	Description
<b>Inverted Nibble</b>	<p>Sets the value of the inverted nibble data on which to trigger.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Fast Channel</b> and the bus is set up for <b>1</b> fast data channel and Data Nibbles is set to <b>6 Secure</b>.</p>
<b>Message ID</b>	<p>Sets the value of the message ID data on which to trigger.</p> <p>Available when <b>Trigger On = Slow Channel</b>.</p>
<b>Data</b>	<p>Sets the condition and value of the slow channel data on which to trigger.</p> <p>Tap the down arrow and select the condition on which to trigger (=, ≠, &gt;, &lt;, ≥, ≤). The default is =.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Slow Channel</b>.</p>
<b>Trigger When</b>	<p>Sets the trigger when condition.</p> <p>Tap the down arrow and select the condition on which to trigger (=, ≠, &gt;, &lt;, ≥, ≤, <b>Inside Range</b>, <b>Outside Range</b>). The default is =.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set high and low boundary values for the number of clock ticks on which to trigger.</p> <p>Available when <b>Trigger On = Pause Pulse</b>.</p>
<b>Number of Ticks</b>	<p>Sets the number of Pause Pulse ticks on which to trigger.</p> <p>Tap the <b>Ticks High</b> or <b>Tick Low</b> field and use the <b>A</b> and <b>B</b> knobs to set the values.</p> <p>Available when <b>Trigger On = Pause Pulse</b> and <b>Trigger When</b> is set to =, ≠, &gt;, &lt;, ≥, or ≤.</p>
<b>Ticks High, Ticks Low)</b>	<p>Sets the highest and lowest range of the number of Pause Pulse ticks on which to trigger.</p> <p>Tap the <b>Ticks High</b> or <b>Tick Low</b> field and use the <b>A</b> and <b>B</b> knobs to set the values.</p> <p>Available when <b>Trigger On = Pause Pulse</b> and <b>Trigger When = Inside Range</b> or <b>Outside Range</b>.</p>
<b>Error Type</b>	<p>Sets the error type on which to trigger. Tap the arrow and select the error condition.</p> <p>Available when <b>Trigger On = Error</b>.</p>

Field or control	Description
<b>CRC Type</b>	<p>Sets the CRC error type on which to trigger (<b>Fast Channel</b> or <b>Slow Channel</b>).</p> <p>Available when <b>Trigger On = Error</b> and <b>Error Type = CRC</b>.</p>

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>



**SPI serial bus trigger settings panel.** Requires option SREMBD. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>SPI</b> bus on which you want to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Data Words</b>	Sets the number of data words (1 word = 8 bits) on which to trigger (one to sixteen bytes). Use the <b>A</b> knob to change the value.  Available when <b>Trigger On = Data</b> .
<b>Data</b>	Sets the data pattern on which to trigger. The number of bits shown depends on the <b>Data Words</b> setting.  Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Data</b> .
<b>A, B knob controls</b>	Use the <b>A</b> knob to select (highlight) the digit(s) to change.  Use the <b>B</b> knob to change the value of the digit(s).

Field or control	Description
<b>Trigger Mode</b>	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:  <b>Auto</b> 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.  <b>Normal</b> 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.
<b>Force Trigger</b>	Forces a trigger event regardless of whether the waveform meets any trigger conditions.

Field or control	Description
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**SPMI serial bus trigger settings panel.** Requires option SRPM. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>SPMI</b> bus on which to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Slave Address</b>	<p>Sets the slave address value on which to trigger.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Reset, Authenticate, Register Read, Register Write, Extended Register Read, Extended Register Write, Ext. Register Read Long, Ext. Register Write Long, DD Block Slave Read, or Register 0 Write.</b></p>
<b>Authenticate</b>	<p>Sets the authentication condition on which to trigger.</p> <p>Available when <b>Trigger On = Authenticate.</b></p>
<b>Master Address</b>	<p>Sets the master address value on which to trigger.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Master Read, Master Write, or DD Block Master Read.</b></p>
<b>Register Address</b>	<p>Sets the register address value on which to trigger.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Master Read, Master Write, Register Read, Register Write, Extended Register Read, Extended Register Write, Ext. Register Read Long, or Ext. Register Write Long.</b></p>
<b>Data</b>	<p>Sets the data value on which to trigger.</p> <p>Tap the <b>Binary</b> or <b>Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Master Read, Master Write, Register Read, Register Write, Extended Register Read, Extended Register Write, Ext. Register Read Long, Ext. Register Write Long, DD Block Master Read, DD Block Slave Read, or Register 0 Write.</b></p>

Field or control	Description
<b>Data Bytes</b>	<p>Sets the number of data bytes on which to trigger. Tap the field and use the <b>A</b> knob to change the value. Or double-tap on the field and use the virtual keypad to enter a value.</p> <p>Available when <b>Trigger On = Extended Register Read, Extended Register Write, Ext. Register Read Long, or Ext. Register Write Long.</b></p>
<b>No Response</b>	<p>Sets to trigger on data that is all zeros (no response). All values in the Data field are set to zero and cannot be edited.</p> <p>Available when <b>Trigger On = Master Read, Register Read, Extended Register Read, Ext. Register Read Long, DD Block Master Read, DD Block Slave Read, or Transfer Bus Ownership.</b></p>

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>

---

Field or control	Description
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**USB serial bus trigger settings panel.** Requires option SRUSB2. Mode & Holdoff panel (Bus Trigger configuration panel) fields and controls

Field or control	Description
<b>Source</b>	Select the <b>USB</b> bus on which to trigger.
<b>Trigger On</b>	Select the type of information on which to trigger.
<b>Handshake Type</b>	Sets the handshake packet type on which to trigger.  Available when <b>Trigger On = Handshake Packet</b> .
<b>Packet Type</b>	Sets the special packet type on which to trigger.  Available when <b>Trigger On = Special Packet</b> .
<b>Error Type</b>	Sets the error type on which to trigger.  Available when <b>Trigger On = Error</b> .
<b>Address</b>	Sets the token packet address pattern on which to trigger. Use in conjunction with the <b>Trigger When</b> field to specify the exact trigger condition.  Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Token Packet</b> and <b>Token Type = all</b> except SOF (0101).
<b>Address Low, Address High</b>	Sets the boundary address conditions when testing for in-range or out of range conditions.  Available when <b>Trigger On = Token Packet</b> and <b>Trigger When = Inside Range</b> or <b>Outside Range</b> .
<b>Data Low, Data High</b>	Sets the boundary data conditions when testing for in-range or out of range conditions.  Available when <b>Trigger On = Data Packet</b> and <b>Trigger When = Inside Range</b> or <b>Outside Range</b> .
<b>Token Type</b>	Sets the token packet type on which to trigger.  Available when <b>Trigger On = Token Packet</b> .
<b>Endpoint</b>	Sets the token packet endpoint pattern on which to trigger.  Use in conjunction with the <b>Trigger When</b> field to specify the exact trigger condition. Tap the <b>Binary</b> , <b>Hex</b> , or <b>Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.  Available when <b>Trigger On = Token Packet</b> and <b>Token Type = all</b> except SOF (0101).

Field or control	Description
<b>Frame Number</b>	<p>Sets the frame number pattern on which to trigger. Use in conjunction with the <b>Trigger When</b> field to specify the exact trigger condition.</p> <p>Tap the <b>Binary, Hex, or Decimal</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Token Packet</b> and <b>Token Type = SOF (0101)</b>.</p>
<b>Data Packet Type</b>	<p>Sets the data packet type on which to trigger.</p> <p>Available when <b>Trigger On = Data Packet</b>.</p>
<b>Data Bytes</b>	<p>Sets the number of data bytes on which to trigger (one to two bytes). Tap the field and use the <b>A</b> knob to change the value.</p> <p>Available when <b>Trigger On = Data Packet</b>.</p>
<b>Byte Offset</b>	<p>Sets the byte offset (<b>Don't Care</b> or the number of bytes). Tap the field and use the <b>A</b> knob to change the value.</p> <p>Available when <b>Trigger On = Data Packet</b>.</p>
<b>Data</b>	<p>Sets the data packet pattern on which to trigger. The number of bits shown depends on the <b>Data Bytes</b> setting. Use in conjunction with the <b>Trigger When</b> field to specify the exact trigger condition.</p> <p>Tap the <b>Binary, Hex, or ASCII</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Data Packet</b> and <b>Trigger When = anything but Inside Range or Outside Range</b>.</p>
<b>Trigger When</b>	<p>Sets the trigger when condition.</p> <p>When set to <b>Inside Range</b> or <b>Outside Range</b>, fields are displayed to set a high and low boundary pattern for the specified trigger type.</p> <p>Tap the <b>Binary or Hex</b> field and use the <b>A</b> and <b>B</b> knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.</p> <p>Available when <b>Trigger On = Token Packet or Data Packet</b>.</p>
<b>A, B knob controls</b>	<p>Use the <b>A</b> knob to select (highlight) the digit(s) to change.</p> <p>Use the <b>B</b> knob to change the value of the digit(s).</p>

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>



## 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
<b>Use Clock Edge?</b>	Enables or disables finding logic patterns that occur on the specified clock edge.
<b>Logic Pattern Define Inputs</b>	Opens the <b>Logic Trigger - Define Inputs</b> menu where you define the logic state ( <b>High</b> , <b>Low</b> , or <b>Don't Care</b> ), and the signal threshold level that defines the logic state (high or low), for each analog or digital signal. See <a href="#">Logic Trigger - Define Inputs configuration menu</a> on page 529.
<b>Trigger When</b> (Use Clock Edge = No)	Defines the waveform condition on which to trigger. <ul style="list-style-type: none"> <li>■ <b>Goes True:</b> All conditions change to a true state.</li> <li>■ <b>Goes False:</b> All conditions change to a false state.</li> <li>■ <b>Is True &gt; Limit:</b> Condition remains true longer than a specified time.</li> <li>■ <b>Is True &lt; Limit:</b> Condition remains true for less than a specified time.</li> <li>■ <b>Is True = Limit:</b> Condition remains true for a specified time (within <math>\pm 5\%</math>).</li> <li>■ <b>Is True <math>\neq</math> Limit:</b> Condition does not remain true for a specified time (within <math>\pm 5\%</math>).</li> </ul>
<b>Clock Source</b> (Use Clock Edge = Yes)	Sets the signal to use as the clock. The clock signal can be a digital, analog, or math waveform
<b>Clock Edge</b> (Use Clock Edge = Yes)	Sets the signal transition edge (rising, falling, or either) for evaluating the logic condition at that clock transition.
<b>Clock Threshold</b> (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).
<b>Define Logic</b>	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. <ul style="list-style-type: none"> <li>■ <b>AND:</b> All conditions are true.</li> <li>■ <b>OR:</b> Any condition is true.</li> <li>■ <b>NAND:</b> One or more conditions are true.</li> <li>■ <b>NOR:</b> No conditions are true.</li> </ul>

**Mode & Holdoff panel (Logic Trigger configuration menu) - fields and controls.**

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**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
<b>Chx (analog channels) or Dx (digital channels)</b>	<p>Use to select the signal sources logic condition on which to perform the logic search (<b>High, Low, Don't Care</b>). Tap to select.</p> <p>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.</p> <p>Use the <b>Threshold</b> field to set the signal level that must be exceeded for that signal to be true (logical 1).</p>
<b>Set All</b>	Sets all signal sources to detect a logic <b>High, Low, or Don't Care</b> 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
<b>Source</b>	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.
<b>Trigger When</b>	<ul style="list-style-type: none"> <li>■ <b>&lt; Limit:</b> A pulse width is less than the specified time limit.</li> <li>■ <b>&gt; Limit:</b> A pulse width is greater than the specified time limit.</li> <li>■ <b>= Limit:</b> A pulse width is equal to the specified time limit.</li> <li>■ <b>≠ Limit:</b> A pulse width does not equal (is greater than or less than) the specified time limit.</li> <li>■ <b>Inside Range:</b> A pulse width is in the specified time range.</li> <li>■ <b>Outside Range:</b> A pulse width is outside of the specified time range.</li> </ul>
<b>Level</b>	Sets the amplitude level that the signal must pass through to be considered a valid transition.
<b>Set to 50%</b>	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
<b>Time Limit</b>	Sets the time period condition to be met.
<b>High Time Limit</b> (Trigger When = Inside Range or Outside Range)	Sets the longest acceptable pulse width time period for the range condition.
<b>Low Time Limit</b> (Trigger When = Inside Range or Outside Range)	Sets the shortest acceptable pulse width time period for the range condition.
<b>Polarity</b>	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
<b>Logic Qualification</b>	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.
<b>Define Inputs</b>	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 <a href="#">Logic Qualification - Define Inputs configuration menu</a> on page 548. Available when Logic Qualification = On.

**Mode & Holdoff panel (Pulse Width Trigger configuration menu) fields and controls.**

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**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
<b>Source</b>	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.
<b>Trigger When</b>	<ul style="list-style-type: none"> <li>■ <b>&lt; Limit:</b> A signal has a rise/fall time less than the specified time limit.</li> <li>■ <b>&gt; Limit:</b> A signal has a rise/fall time greater than the specified time limit.</li> <li>■ <b>= Limit:</b> A signal has a rise/fall time that is equal to the specified time limit (<math>\pm 5\%</math>).</li> <li>■ <b><math>\neq</math> Limit:</b> A signal has a rise/fall time that does not equal (is greater than or less than) the specified time limit (<math>\pm 5\%</math>).</li> </ul>
<b>Time Limit</b>	Sets the time period condition to be met.
<b>Slope</b>	Sets the signal transition direction to detect. (rising, falling, or either direction).
<b>Upper Threshold</b>	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
<b>Lower Threshold</b>	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
<b>Logic Qualification</b>	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.
<b>Define Inputs</b>	Opens the <b>Logic Qualification - Define Inputs</b> configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See <a href="#">Logic Qualification - Define Inputs configuration menu</a> on page 548. Available when Logic Qualification = On.

**Mode & Holdoff panel (Rise/Fall Time Trigger configuration menu) fields and controls.**

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>



**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
<b>Source</b>	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.
<b>Trigger When</b>	<ul style="list-style-type: none"> <li>■ <b>Occurs:</b> A runt signal event occurs.</li> <li>■ <b>&lt; Limit:</b> A runt signal event occurs that has a pulse width less than the specified time limit.</li> <li>■ <b>&gt; Limit:</b> A runt signal event occurs that has a pulse width greater than the specified time limit.</li> <li>■ <b>= Limit:</b> A runt signal event occurs that has a pulse width that is equal to the specified time limit (<math>\pm 5\%</math>).</li> <li>■ <b><math>\neq</math> Limit:</b> A runt signal event occurs that has a pulse width that does not equal (is greater than or less than) the specified time limit (<math>\pm 5\%</math>).</li> </ul>
<b>Polarity</b>	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
<b>Time Limit</b>	Sets the time period condition to be met.
<b>Upper Threshold</b>	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
<b>Lower Threshold</b>	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
<b>Logic Qualification</b>	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.
<b>Define Inputs</b>	Opens the <b>Logic Qualification - Define Inputs</b> configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See <a href="#">Logic Qualification - Define Inputs configuration menu</a> on page 548. Available when Logic Qualification = On.

**Mode & Holdoff panel (Runt Trigger configuration menu) fields and controls.**

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	Tap the <b>Holdoff Time</b> 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.
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**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
<b>A Trigger Event</b>	<p>Tap to open the <b>A Trigger Event</b> menu to select the first (A) event trigger condition. See <a href="#">A Trigger Event configuration menu</a> on page 541.</p> <p>If the A event does not occur, no trigger event is generated.</p> <p>The <b>Sequence</b> trigger type is not available in the A Trigger Event menu.</p>
<b>B Trigger Event</b>	<p>Tap to open the <b>B Trigger Event</b> menu to select the second (B) event trigger condition. See <a href="#">B Trigger Event configuration menu</a> on page 541.</p> <p>If the A event occurs but the B event does not occur, no trigger event is generated.</p> <p>The <b>Sequence</b> trigger type is not available in the B Trigger Event menu.</p>
<b>After the A Trigger Event is found: Trigger on the 1st B event</b>	<p>Sets the oscilloscope to trigger on the first occurrence of the B event trigger conditions.</p>
<b>After a Delay of:</b>	<p>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.</p> <p>Available when After the A Trigger Event is found = Trigger on the 1st B event.</p>
<b>After the A Trigger Event is found: Trigger on the Nth B event</b>	<p>Sets the B trigger event to wait for a specified number of trigger events before generating a trigger.</p>
<b>Where N is:</b>	<p>Sets the number of B trigger events that must occur before triggering the oscilloscope.</p> <p>Available when After the A Trigger Event is found = Trigger on the Nth B event.</p>

**Mode & Holdoff panel (Sequence Trigger configuration menu) fields and controls.**

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
<b>Holdoff</b>	Holdoff is not compatible with Sequence triggering.
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**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
<b>Trigger Type</b>	Select the A event trigger type.
<b>Other fields and controls</b>	<p>The fields and controls shown depend on the selected Trigger Type. Use the following links to access setting information on a specific trigger.</p> <ul style="list-style-type: none"> <li>■ <a href="#">Bus Trigger Menu</a></li> <li>■ <a href="#">Edge Trigger menu</a></li> <li>■ <a href="#">Logic Trigger</a></li> <li>■ <a href="#">Pulse Width Trigger menu</a></li> <li>■ <a href="#">Rise/Fall Time Trigger menu</a></li> <li>■ <a href="#">Runt Trigger menu</a></li> <li>■ <a href="#">Sequence Trigger menu</a></li> <li>■ <a href="#">Setup &amp; Hold Trigger menu</a></li> <li>■ <a href="#">Timeout Trigger menu</a></li> <li>■ <a href="#">Window Trigger menu</a></li> </ul> <p><b>NOTE.</b> The Sequence Trigger type is not available in the A Trigger Event menu.</p>

**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
<b>Trigger Type</b>	Select the B event trigger type.
<b>Other fields and controls</b>	<p>The fields and controls shown depend on the selected Trigger Type. Use the following links to access setting information on a specific trigger.</p> <ul style="list-style-type: none"> <li>■ <a href="#">Bus Trigger Menu</a></li> <li>■ <a href="#">Edge Trigger menu</a></li> <li>■ <a href="#">Logic Trigger</a></li> <li>■ <a href="#">Pulse Width Trigger menu</a></li> <li>■ <a href="#">Rise/Fall Time Trigger menu</a></li> <li>■ <a href="#">Runt Trigger menu</a></li> <li>■ <a href="#">Sequence Trigger menu</a></li> <li>■ <a href="#">Setup &amp; Hold Trigger menu</a></li> <li>■ <a href="#">Timeout Trigger menu</a></li> <li>■ <a href="#">Window Trigger menu</a></li> </ul> <p><b>NOTE.</b> The Sequence Trigger type is not available in the A Trigger Event menu.</p>

**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
<b>Clock Source</b>	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform.
<b>Clock Level</b>	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).
<b>Clock Edge</b>	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.
<b>Data Sources: Define Inputs</b>	Opens the Setup & Hold Trigger - Define Inputs menu. Use this menu to select the input signals and their thresholds. See <a href="#">Setup and Hold Trigger - Define Inputs configuration menu</a> on page 545.
<b>Setup Time</b>	Sets the length of time that data signal should be stable and not change before a clock edge occurs.
<b>Hold Time</b>	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
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**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
<b>Chx (analog channels) or Dx (digital channels)</b>	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.
<b>Set All</b>	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
<b>Source</b>	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.
<b>Trigger When</b>	<ul style="list-style-type: none"> <li>■ <b>Stays High:</b> The signal stays above the specified threshold level longer than the specified time.</li> <li>■ <b>Stays Low:</b> The signal stays below the specified threshold level longer than the specified time.</li> <li>■ <b>Either:</b> The signal stays above or below the specified threshold level longer than the specified time.</li> </ul>
<b>Threshold</b>	Sets the amplitude level that the signal must pass through to be considered a valid transition.
<b>Set to 50%</b>	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as $(\text{Top} + \text{Bottom})/2$ .
<b>Time Limit</b>	Sets the time period condition to be met.
<b>Logic Qualification</b>	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.
<b>Define Inputs</b>	<p>Opens the <b>Logic Qualification - Define Inputs</b> configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See <a href="#">Logic Qualification - Define Inputs configuration menu</a> on page 548.</p> <p>Available when Logic Qualification = On.</p>

**Mode & Holdoff panel (Timeout Trigger configuration menu) fields and controls.**

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**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
<b>Chx (analog channels) or Dx (digital channels)</b>	<p>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.</p> <p>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.</p> <p>Use the Threshold field to set the signal level that must be exceeded for that signal to be true (logical 1).</p>
<b>Set All</b>	Sets all signal sources to detect a logic <b>High</b> , <b>Low</b> , or <b>Don't Care</b> condition.
<b>Define Logic</b>	<p>Sets the logic condition that must occur with all inputs.</p> <ul style="list-style-type: none"> <li>■ <b>AND</b>: All conditions are true.</li> <li>■ <b>OR</b>: Any condition is true.</li> <li>■ <b>NAND</b>: One or more conditions are true.</li> <li>■ <b>NOR</b>: No conditions are true.</li> </ul>

**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
<b>Source</b>	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.
<b>Trigger When</b>	<ul style="list-style-type: none"> <li>■ <b>Enters Window:</b> The signal outside a window enters the window defined by the upper and lower threshold settings.</li> <li>■ <b>Exits Window:</b> The signal exits the window defined by the upper and lower threshold settings.</li> <li>■ <b>Inside &gt; Limit:</b> The signal remains inside the window longer than the specified time limit.</li> <li>■ <b>Outside &gt; Limit:</b> The signal remains outside the window longer than the specified time limit.</li> </ul>
<b>Upper Threshold</b>	Sets the amplitude threshold value for the upper edge of the window.
<b>Lower Threshold</b>	Sets the amplitude threshold value for the lower edge of the window.
<b>Time Limit</b>	<p>Sets the time period condition to be met.</p> <p>Available when Trigger When = Inside &gt; Limit or Outside &gt; Limit.</p>
<b>Threshold Crossing</b> (Trigger When = Outside > Limit)	<ul style="list-style-type: none"> <li>■ <b>Upper:</b> 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.</li> <li>■ <b>Lower:</b> 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.</li> <li>■ <b>Either:</b> A signal remains outside (above or below) the two threshold levels for longer than the specified time limit before crossing either threshold level.</li> <li>■ <b>None:</b> A signal remains outside the two specified threshold levels for longer than a specified time limit.</li> </ul>
<b>Threshold Crossing</b> (Trigger When = Inside > Limit)	<ul style="list-style-type: none"> <li>■ <b>Upper:</b> A signal remains between two thresholds for longer than the specified time limit before crossing through the upper threshold.</li> <li>■ <b>Lower:</b> A signal remains between two thresholds for longer than the specified time limit before crossing through the lower threshold.</li> <li>■ <b>Either:</b> A signal remains between two thresholds for longer than the specified time limit before crossing through either the upper or lower threshold.</li> <li>■ <b>None:</b> A signal remains between two threshold levels for longer than a specified time limit.</li> </ul>
<b>Logic Qualification</b>	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.



---

Field or control	Description
<b>Define Inputs</b>	<p>Opens the <b>Logic Qualification - Define Inputs</b> configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See <a href="#">Logic Qualification - Define Inputs configuration menu</a> on page 548.</p> <p>Available when Logic Qualification = On.</p>

**Mode & Holdoff panel (Window Trigger configuration menu) fields and controls.**

Field or control	Description
<b>Trigger Mode</b>	<p>The trigger mode determines how the instrument behaves in the absence or presence of a trigger event:</p> <p><b>Auto</b> 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.</p> <p>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.</p> <p>If valid triggers occur, the display will become stable.</p> <p><b>Normal</b> 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.</p>
<b>Force Trigger</b>	<p>Forces a trigger event regardless of whether the waveform meets any trigger conditions.</p>
<b>Holdoff</b>	<p>Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event.</p> <p><b>Random</b> 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.</p> <p><b>Time</b> 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.</p>
<b>Holdoff Time</b>	<p>Tap the <b>Holdoff Time</b> 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.</p>
<b>Trigger Frequency Counter</b>	<p>Turn <b>On</b> to display the trigger event frequency in the <b>Trigger</b> badge.</p> <p>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.</p> <p>Only available if you have installed the DVM option, which is available when you register your instrument with Tektronix.</p>

**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](#)

**Visual Trigger panel**

Use the Visual Trigger panel in the Trigger badge to enable Visual Trigger mode and set area combinatorial logic.

To access the **Visual Trigger** panel:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Tap the **Visual Trigger** panel.

**Visual Trigger panel (Trigger configuration menu) fields and controls.**

Field or control	Description
<b>Visual Trigger</b>	Enables or disables Visual Trigger mode
<b>Display</b>	<p><b>Areas:</b> When this is checked, draws the visual trigger areas in the Waveform View.</p> <p><b>Criteria:</b> When this is checked, the text associated with areas is displayed (for example, the area name (A1), the criteria symbol (in, out, don't care), and the source (in overlay)).</p> <p><b>Logic Equation:</b> When this is checked, the Area Combinatorial Logic equation is displayed in white text in the upper left of the Waveform View. You can move the equation by dragging it to a new position.</p> <p>When zoom is on, the equation is in the upper left of the Zoom Overview area.</p> <p>To create or edit the area combinatorial logic equation, double tap on the equation to open the <b>Area Combinatorial Logic</b> keypad. Note that you can only open the keypad if more than one visual trigger area is defined. See <a href="#">Visual Trigger Area Combinatorial Logic menu</a> on page 558</p>

## 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.

## Bus trigger radix-specific virtual keypads

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

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

### Virtual Keypad fields and controls

Field or control	Description
<b>Clear</b>	Sets all digits to X (don't care) for binary, hex, octal and ASCII formats. Decimal gets set to 0.
<	Moves the insertion point left and highlights the character that can be edited.
>	Moves the insertion point right and highlights the character that can be edited.
Radix keypad	Provides a keypad associated with the logic entry radix field that was double-tapped in the Trigger menu. Tapping a key sets the selected digit to the specified value and moves the selected digit indicator to the next (right) digit.  You can also use an attached keyboard to enter values in the keypad character field.
<b>Enter</b>	Closes the number pad and assigns the entered value to the field. Also updates all other formats in the trigger menu so that all formats are indicating the same value.
<b>Cancel</b>	Closes the logic keypad without saving entered values.

## 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
<b>Clear</b>	Clears all values from the input entry field.
<b>Exp</b>	Lets you enter exponential notation entries.
<b>Max</b>	Enters the maximum value allowed for this setting.
<b>Min</b>	Enters the minimum value allowed for this setting.
<b>Bksp</b> ←	Deletes characters to the left of the insert text marker position.
<b>Enter</b>	Closes the number pad and assigns the entered value to the field.
<b>±</b>	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.

## Visual Trigger Area configuration menu

Use the Visual Trigger Area menu to edit visual trigger area parameters. Double tapping a Visual Trigger area opens the Area configuration menu and makes the related source the selected source.

To open the Visual Trigger **Area** menu, double-tap on a visual trigger area.

To create a visual trigger area, see [Create Visual Trigger areas](#) on page 154

### Area Settings panel fields and controls

Field or control	Description
<b>Source</b>	Lists the available source signals for which to create an area. The default value is the signal source of the selected area.
<b>Shape</b>	Lists the current shape type, and also lets you change the current shape to a specified shape. Changing a shape defines the minimum rectangle that includes all vertices of the current area, and then does a best-fit approach to create the specified shape.  If you made changes to a default shaped area that results in an area that no longer meets the default shape definition, that area is listed as a Custom shape in the menu.

Field or control	Description
<b>Waveform Must Be</b>	<p>Sets the logic condition for how the area logic equation interprets when a waveform intersects an area.</p> <p><b>In (✓)</b>: A waveform must intersect this area for the area logic to be true.</p> <p><b>Out (X)</b>: A waveform must not intersect this area for the area logic to be true.</p> <p><b>Don't Care (?)</b>: Don't care if a waveform intersects the area. This setting lets you ignore an area during visual trigger development, instead of having to delete and recreate areas.</p>
<b>Height</b>	Sets the area height, in amplitude units, between the topmost vertex and the bottommost vertex.
<b>Vertical Center</b>	Sets the area vertical center, in amplitude units, as the point halfway between the topmost vertex and bottommost vertex.
<b>Flip Vertical</b>	Flips the area vertically around its Vertical Center value.
<b>Width</b>	Sets the area width, in time units, between the leftmost vertex and the rightmost vertex.
<b>Horizontal Center</b>	Sets the area horizontal center, as time units, as the point halfway between the leftmost vertex and rightmost vertex.
<b>Flip Horizontal</b>	Flips the area horizontally around its Horizontal Center value.
<b>Rotate</b>	<p>Rotates the area in units of degrees, from 0° to 360°.</p> <p>The rotation is an absolute angle measurement referenced from 0°, where 0° is the position of the area when it was first created. For example, if you rotate the area 40°, and then rotate it again with 20°, the resulting area rotation is 20°.</p> <p>The area height and width are relative to the current area orientation, and automatically change places as needed when the area is rotated.</p>

**Edit Vertices panel fields and controls**

Field or control	Description
<b>Define Area</b>	<p>A table that lists the X (time) and Y (amplitude) values for each vertex (point) of the area. Use the scroll bar to show points if there are more than nine points in the area.</p> <p>Selecting a row in the table highlights the associated vertex on the area.</p> <p>Use the Multipurpose knobs to change the value of the X or Y settings, or double-tap a setting and enter the value directly.</p>
<b>Insert Point</b>	<p>Inserts a new row above the selected row and creates a new vertex on the area shape. The new vertex is halfway between the vertices defined in the prior row and the following row in the table.</p>
<b>Delete Point</b>	<p>Deletes the currently selected point, keeps the row selected, and moves all rows below it up one row.</p> <p>The <b>Delete Point</b> button is not available when a triangular area is selected.</p>
<b>Reset Points</b>	<p>Deletes all but three data points from the table. The remaining three data points are set to a default triangle, set to two divisions for height and width, and centered in the waveform area.</p>

---

**NOTE.** *Redo and Undo are available for most Edit Vertices panel controls.*

---

**Visual trigger right click menu**

See [Right click menu functions associated with visual trigger areas](#) on page 559.

## Visual Trigger Area Combinatorial Logic menu

Use the Area Combinatorial Logic menu to describe the logic conditions required for a true condition of all associated areas.

To open the Visual Trigger **Area Combinatorial Logic** menu:

1. Double-tap the **Trigger** badge.
2. Tap the **Visual Trigger** panel.
3. Double-tap the **Area Combinatorial Logic** field to open the **Area Combinatorial Logic** menu.

---

**NOTE.** *There must be at least two areas before you can open the **Area Combinatorial Logic** menu.*

---

### Area Combinatorial Logic menu fields and controls

Field or control	Description
<b>Logic expression field</b>	<p>Displays the area combination logic for all areas associated with the same signal source. All areas are ANDed by default, including areas set as Don't Care.</p> <p>Use the buttons in the menu to edit or define the area combination logic, or double-tap the field and enter the logic expression directly.</p> <p>You must tap the <b>Enter</b> button to check and save the logic expression to make it valid.</p>
<b>Clear</b>	Deletes all text in the logic expression field.
<b>Move insertion point</b>	Moves the insertion point left or right in the logic expression field.
<b>Logic action buttons</b>	Inserts the selected logic function or grouping parenthesis at the insertion point in the logic expression field.
<b>Bksp</b>	Deletes the character to the left of the insertion point.
<b>Insert Area buttons</b>	<p>Inserts the selected area text into the logic expression. Only available areas are listed.</p> <p>If more than nine areas are defined, use the scroll bar to view area buttons greater than A9.</p>
<b>Cancel</b>	Closes the menu without saving any changes.
<b>Enter</b>	<p>Checks the syntax of the logic expression. If it is valid, the changes are saved and the menu is closed.</p> <p>If the logic expression syntax is not valid, or the logic expression field is empty, the instrument displays an error message, the menu remains open, and no changes are saved.</p>



## Right click menu functions associated with visual trigger areas

The following functions are available when you right click on a visual trigger area.

Field or control	Description
In	Sets the area's Waveform Must Be control to <b>In</b> .
Out	Sets the area's Waveform Must Be control to <b>Out</b> .
Don't Care	Sets the area's Waveform Must Be control to <b>Don't Care</b> .
Rectangle	Sets the area's shape to a rectangle
Triangle	Sets the area's shape to a triangle.
Trapezoid	Sets the area's shape to a trapezoid.
Hexagon	Sets the area's shape to a hexagon.
Create Duplicate	Creates a new area with the same characteristics as the current area, but offset from the original area by X+50 and Y+50.
Configure Area	Opens the Area configuration menu for the selected area.
Delete Area	Deletes the selected area.

## Waveform View configuration menu

Use this menu to configure the Waveform View to set display mode (Stacked or Overlay), 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
<p><b>Display Mode</b></p>	<p>Sets how waveforms are shown on the screen.</p> <p><b>Overlay</b> 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.</p> <p><b>Stacked</b> 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.</p> <p><i><b>NOTE.</b> Grouping badges requires the display mode be set to <b>Stacked</b> to display the grouped waveforms in slices. When grouping badges, the display mode is a combination of Overlay and Stacked modes, where each group of signals is overlaid in a slice for that group.</i></p> <p><i>See the Group badges topic in the Help.</i></p>
<p><b>Interpolation</b></p>	<p>Selects the method used to display record points between sampled points.</p> <p><b>Sin(x)/x</b> 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.</p> <p><b>Linear</b> 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.</p>
<p><b>Persistence</b></p>	<p>Sets the length of time data points are displayed on screen before being erased.</p> <p><b>Off</b> sets the record points to appear for the current acquisition only.</p> <p><b>Infinite</b> 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.</p> <p><b>Variable</b> lets you specify a time length to retain data points on screen. Each record point decays independently according to the time interval.</p> <p><b>Auto</b> sets the <b>Waveform Intensity</b> field to control the persistence time.</p>

Fields or controls	Description
<b>Variable Persistence Time</b> (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.
<b>Waveform Style</b>	Sets how waveforms are drawn on the screen.  <b>Vectors</b> draws waveforms with lines between record points.  <b>Dots</b> draws waveform record points as dots on the screen, and adds crosshair markers to real sampled points.
<b>Waveform Intensity</b>	Sets the brightness of the waveform. Tap the field and use the A knob to set the waveform intensity.
<b>Graticule Style</b>	Sets the type of graticule to display.  <b>Grid</b> shows a traditional grid on the instrument display. This is the default grid mode.  <b>Time</b> displays vertical graticules for just the horizontal (time) scale units.  <b>Full</b> displays the same style graticule lines for both horizontal and vertical scale units  <b>None</b> turns off the graticule, including the vertical scale readouts.
<b>Graticule Intensity</b>	Sets the brightness of the graticule. Tap the field and use the A knob to set the graticule intensity.



# Waveform acquisition concepts

## Acquisition concepts

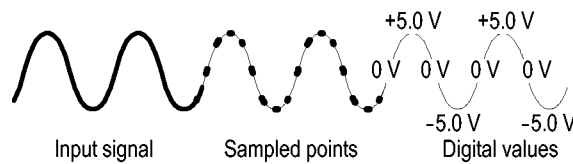
The Acquisition system sets which data points are used to acquire waveforms.

### 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.

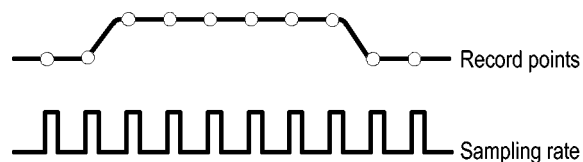
### Sampling process

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.



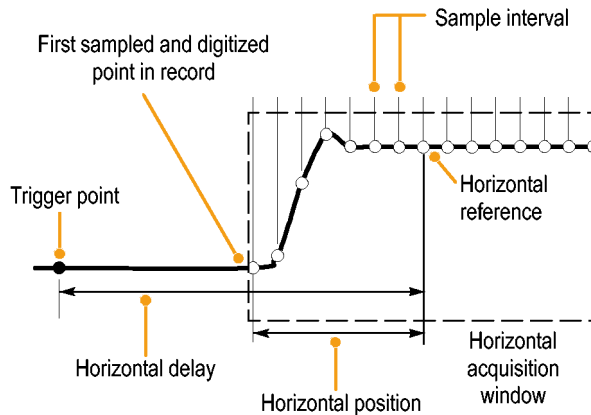
### Interpolated Real-Time sampling

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

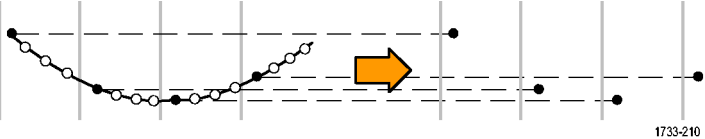
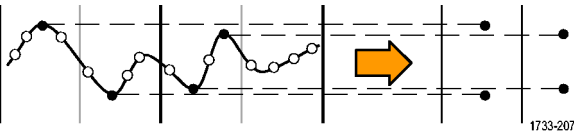
Your instrument can interpolate between the samples it acquires when it does not have all of the actual samples it needs to fill the waveform record. Linear interpolation computes record points between actual acquired samples by using a straight line fit.

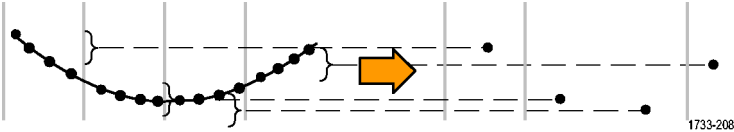


Sin(x)/x interpolation computes record points using a curve fit between the actual values acquired. Sin(x)/x interpolation is the default interpolation mode because it requires fewer actual sample points than linear interpolation to accurately represent the waveform.

# 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	
<p><b>Sample mode</b> 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.</p>	 <p>The diagram shows a waveform with several acquisition intervals marked by vertical dashed lines. In each interval, the first sampled point (represented by a black dot) is retained, while the rest of the samples (represented by white circles) are discarded. An orange arrow points from the first interval to the second, indicating the sequence of acquisition. The reference number 1733-210 is located in the bottom right corner of the diagram.</p>
<p><b>Peak Detect mode</b> retains the highest and lowest values of all the samples in each acquisition interval. This mode only works with real-time, noninterpolated sampling and is useful for catching high frequency glitches.</p>	 <p>The diagram shows a waveform with several acquisition intervals marked by vertical dashed lines. In each interval, the highest and lowest values of all samples (represented by black dots) are retained. An orange arrow points from the first interval to the second, indicating the sequence of acquisition. The reference number 1733-207 is located in the bottom right corner of the diagram.</p>

Acquisition mode	
<p><b>High Res</b> 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.</p>	
<p><b>Envelope</b> mode finds the highest and lowest record points over many acquisitions. Envelope uses Peak Detect for each individual acquisition.</p>	
<p><b>Average</b> 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.</p>	



Acquisition mode	
<p><b>FastAcq™</b> 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.</p>	
<p><b>FastFrame™</b> 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.</p>	
<p><b>Roll Mode</b> scrolls sequential waveform points across the display in a right-to-left rolling motion. Roll mode starts automatically when the timebase is set to <math>\geq 40</math> ms/div. Roll mode works at sample rates up to <b>10 MS/s</b>.</p>	

## 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).

---

**NOTE.** *Enabling FastFrame mode disables FastAcq mode (if it was enabled). Likewise, enabling FastAcq mode disables FastFrame mode (if it was enabled).*

---

## 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 M $\Omega$  or 50  $\Omega$ . 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.

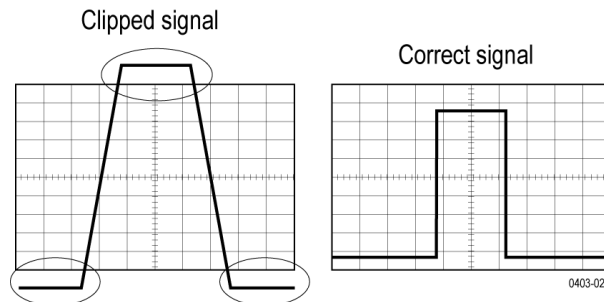
Consider the following when you use 50  $\Omega$  termination with any channel:

- 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  $\Omega$  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 571.



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 572.

---

**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.

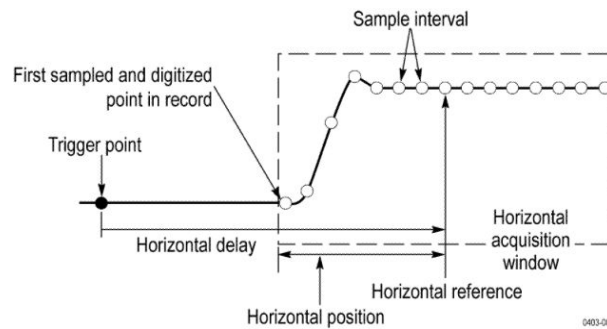
## Horizontal acquisition considerations

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.

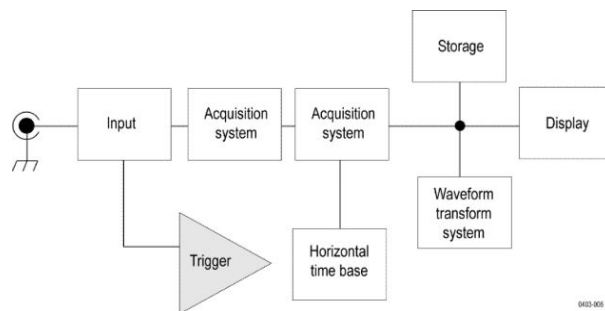


# Trigger concepts

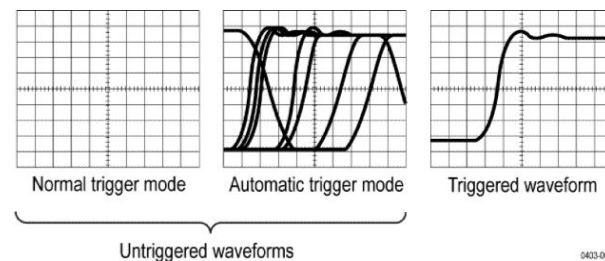
## Triggering concepts

**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). When the edge trigger source is set to **AC Line**, the instrument uses the instrument's AC power line signal to generate a trigger based on the power line frequency. The 6MSO series and the MSO58LP instruments also have an **AUX In** connector that is available as another source for the Edge trigger.

**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 enters or leaves a window set by two threshold levels. You can further qualify the trigger event in terms of time by using the **Trigger When** controls, or by using the logical state of other channels by using the **Logic Qualification** control.

**Logic.** These are special-purpose triggers that are primarily used with digital logic signals. Logic triggers are available on the main (A) 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 and serial buses. A bus trigger event occurs when the instrument detects a bus pattern that you set in a bus trigger configuration menu. You must add a bus to the Waveform view before you can set the trigger parameters for that bus.

## Trigger modes

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

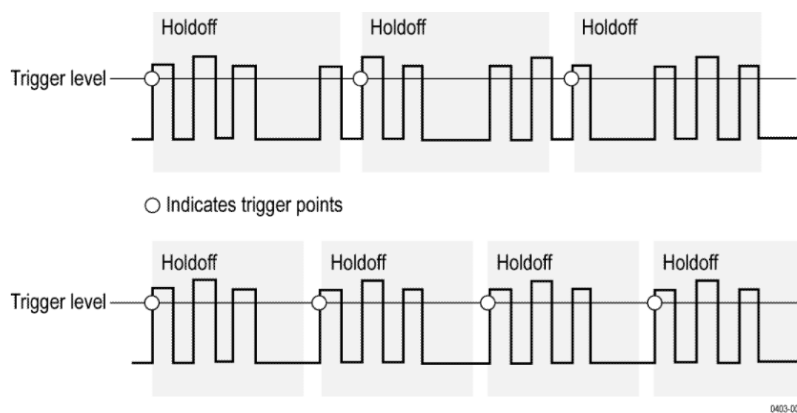
- **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the instrument does not acquire a waveform, and the last waveform record acquired remains "frozen" on the display. If no last waveform exists, no waveform is displayed.
- **Auto** trigger 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 153. 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.

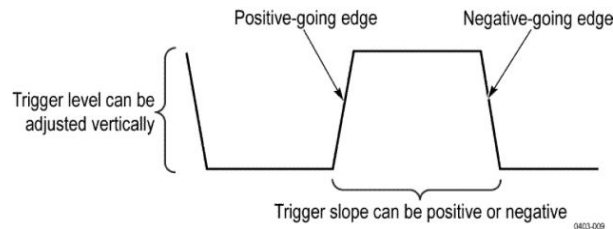
## Trigger coupling

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.

A Trigger	Delay	B Trigger
2 / 1.52 V	79.04 ns	6 / 0 V

## 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.

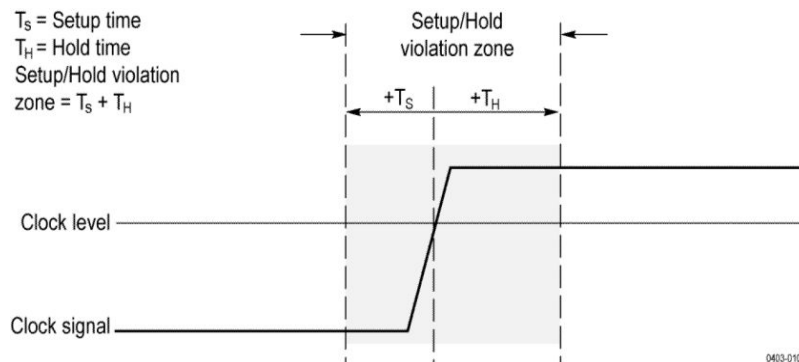
After setting these levels, you can specify whether you want to trigger the instrument as the signal is entering or leaving the threshold window. You can further qualify the trigger event in terms of time, or by the logical state of other channels.

**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.*

---



## Visual Trigger concepts

Visual Trigger makes the identification of the desired waveform events quick and easy by scanning through all acquired analog waveforms and graphically comparing them to geometric shapes on the display. By discarding acquired waveforms that do not meet the graphical definition, Visual Triggering extends the oscilloscope's trigger capabilities beyond the traditional hardware trigger system.

Although Visual Trigger is similar in appearance to mask testing, where acquired waveforms are graphically compared to mask regions on the display, there is an important difference. Visual Trigger only displays waveforms that conform to the specified shape, so only conforming waveforms are measured.

Creating a visual trigger always begins with setting up the oscilloscope's hardware trigger system to acquire the waveforms in a standard trigger mode. The trigger can be as simple as an edge trigger or as complex as a pulse width, runt, logic multi-state trigger.

Once a standard trigger is set up, create visual trigger areas to refine the trigger condition. Each visual trigger area is associated with a specific analog input channel, and you can define areas for multiple channels. See [Create Visual Trigger areas](#) on page 154. By default, newly-created areas are rectangular, are associated with the selected channel, and are assigned the logic condition In when created.

Once areas are created on the oscilloscope's display, they can be re-positioned, re-sized, and/or edited interactively on the screen to create the required shape to define the visual trigger condition. See [Edit visual trigger areas using the Area menu](#) on page 158.

The visual trigger **Area** menu enables more precise editing of each Visual Trigger area. You can set exact coordinates for the vertices of each area in amplitude and time values, and assign the area to a specified signal source (channel). The menu also lets you set the logic condition for each shape (the waveform must be inside the area, outside the area, or don't care). Double-tap on an area to open the Area menu. See [Edit visual trigger areas using the Area menu](#) on page 158.

Finally, a logical equation is automatically generated when two or more areas are added to the display. This equation describes how Visual Trigger uses the areas to determine which acquired waveforms are displayed and which are discarded. For example, the equation  $A1 \& A2 \& A3$  specifies that the waveform must meet the area logic condition for all three areas to display a waveform.

**Visual trigger areas.** How an area is shown depends on the instrument display mode. In Stacked mode, areas are outlined in cyan and have a semi-transparent cyan fill. The areas display the area number and the logic condition setting icon. Areas in Stacked mode do not include a channel label.

In Overlay mode, areas are outlined in the source channel's color and have a semi-transparent fill in the same color. The areas display the area number, the logic condition setting icon, and the channel with which the area is associated.

When zoom is active, areas are drawn in both the overview and zoomed view.

Area vertices all have specific time and amplitude values associated with them. These values are maintained as you change other controls. This means that the size of the areas change to when you change horizontal or vertical scale settings. The areas will also move up or down with vertical position changes. Any text that is displayed as part of the area is still displayed in the standard size, so you can still easily identify the location visual trigger areas.

If a visual trigger area is off the display, the instrument displays a directional arrow that points to the location of the visual trigger area. If zoom is on, the directional arrows are drawn in both the zoom overview and the zoomed view as needed.

**Moving visual trigger areas.** To move a visual trigger area, just touch and drag the area to a new position.

When dragging a visual trigger area while in Stacked or Overlay display mode, the area source remains the same.

When dragging a visual trigger area while in Stacked display mode, you cannot drag a visual trigger area from one waveform slice to another.

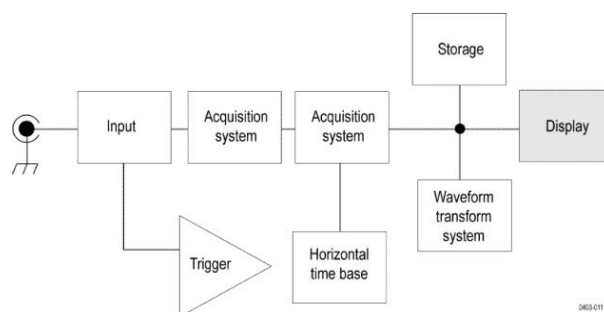
### **Visual trigger caveats.**

- Visual triggering is not compatible with Fast Acq or Fast Frame modes. If visual triggering is active and you turn on one of these modes, then visual triggering is turned off and vice versa.
- You cannot save or recall visual trigger definitions as individual items. However, you can save visual trigger definitions as part of **Session** and **Setup** files.

# 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).

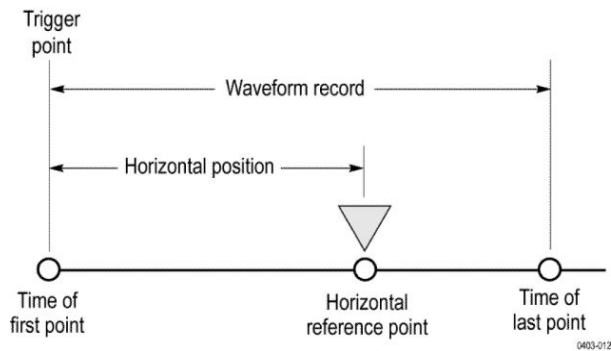
You can also add histogram, spectral, eye, and measurement results views (plots) for individual measurements. These plot views are contained within their own view window and can be moved within the overall view area.

## 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.

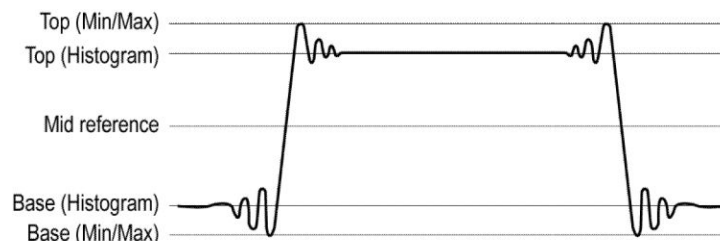
**MinMax** defines the 0% and the 100% waveform levels as the lowest value and the highest value samples of the waveform record. This setting is best for examining waveforms that have no large, flat portions at a common value, such as sine waves and triangle waves - almost any waveform except for pulses.

The MinMax method calculates the Top and Base values as follows:

Top = Max

and

Base = Min



0403-024

**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.

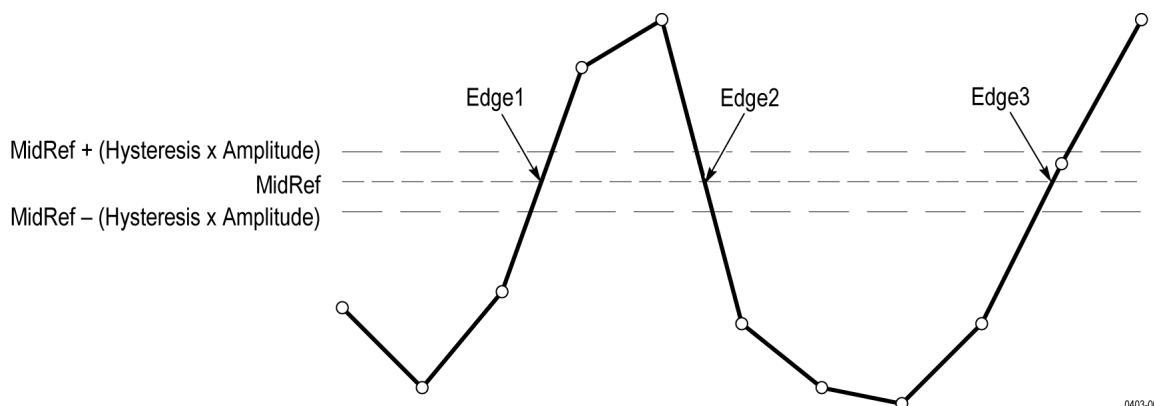
For example, once a crossing has been measured in a negative direction, the waveform data must fall below the hysteresis value of the amplitude from the *Mid* reference point before the measurement system is armed and ready for a positive crossing. Similarly, after a positive *Mid* reference crossing, 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*.

**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.

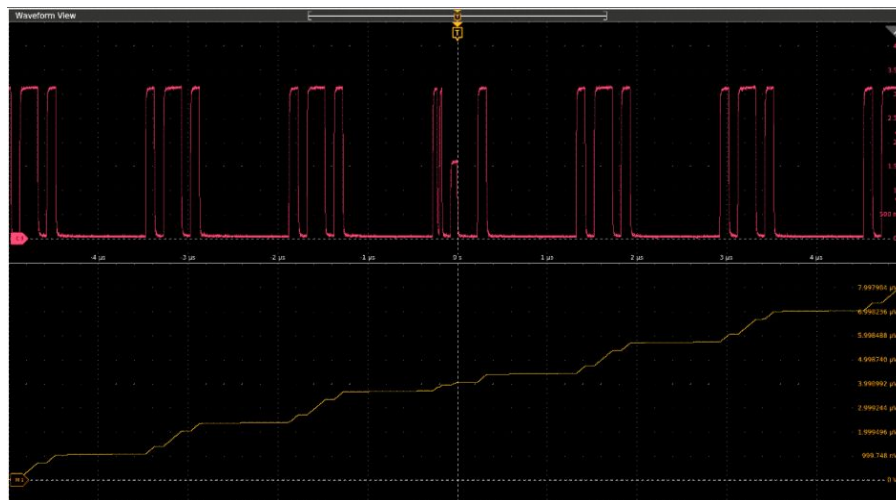
When samples are out of range, the measurement will give a warning to that effect (for example, CLIPPING) if the measurement could change by extending the measurement range slightly. The algorithms assume the samples recover from an overdrive condition instantaneously.

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.

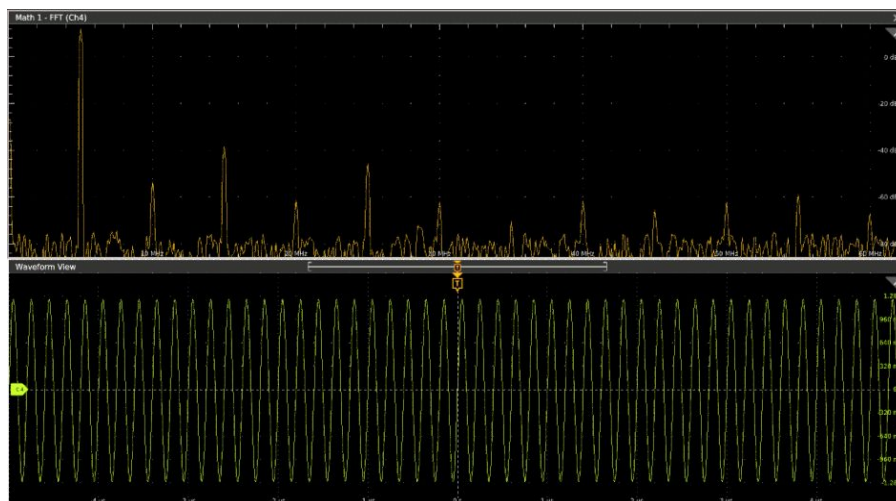


## Math waveforms

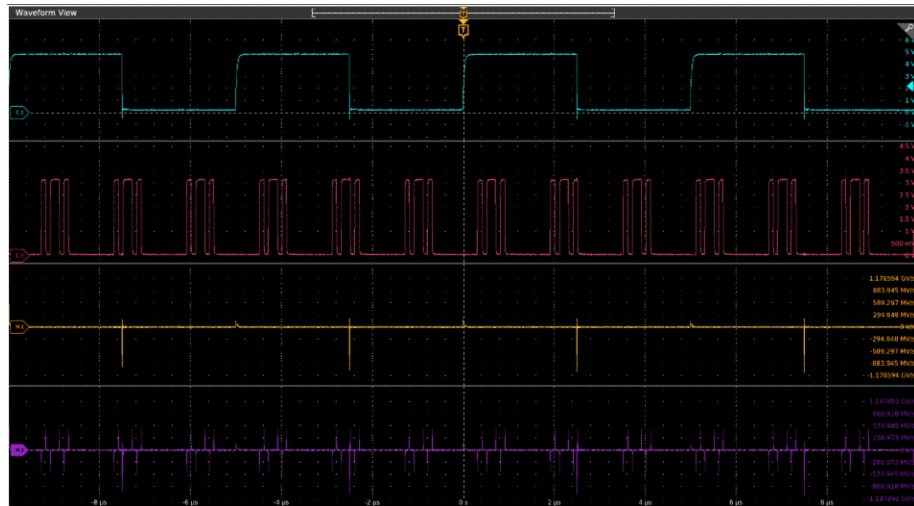
Once you have acquired waveforms or taken measurements on waveforms, the instrument can mathematically combine them to create a waveform that supports your data-analysis task. For example, you might have a waveform clouded by background noise. You can obtain a cleaner waveform by subtracting the background 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.flr"]HighPass(Ch1)

CoefFileName is the setting and is used as the high pass filter on channel 1.

Example: [CoefFileName="highpass\_0.25bw.flr"] HighPass(Ch1) + [CoefFileName="lowpass\_0.05bw.flr"] LowPass(Ch2)

The high pass filter file is applied to channel 1 and lowpass\_0.05bw.flr 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.

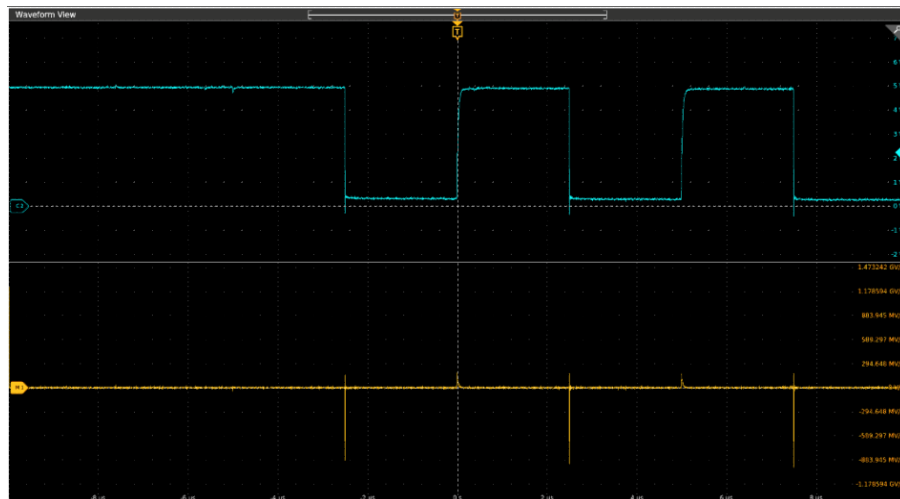
Derivative waveforms are used in the measurement of slew rate of amplifiers and in educational applications. You can create a derivative math waveform and then use it as a source for another derivative waveform. The result is the second derivative of the waveform that was first differentiated.

The math waveform, derived from the sampled waveform, is computed based on the following equation:

$$Y_n = (X(n + 1) - X_n) * 1/T$$

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.

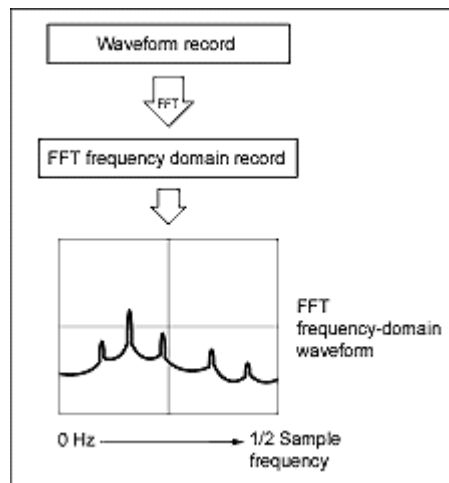
**DC offset**

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 1/2 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.



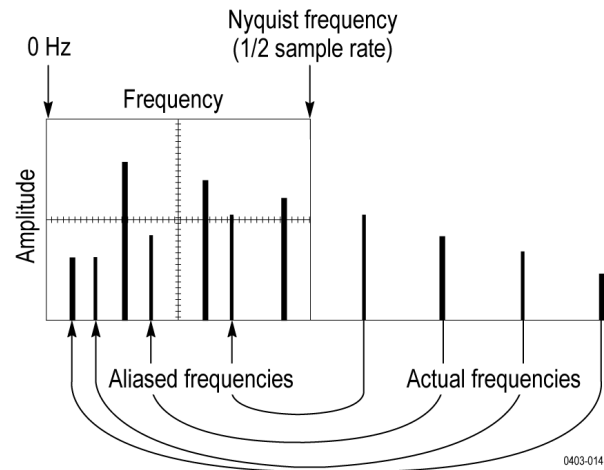
## FFT and aliasing

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.



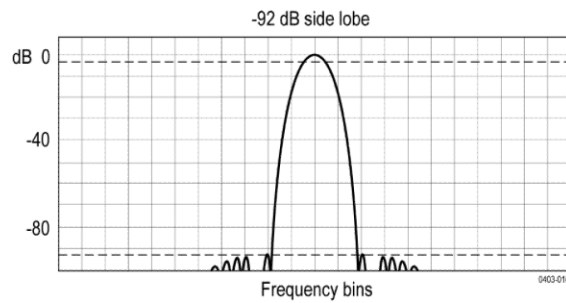
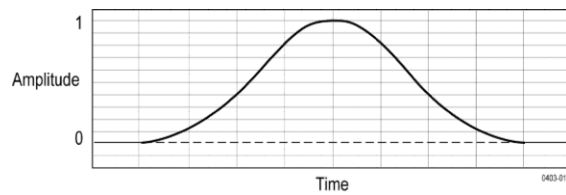
If you have a variable-frequency signal source, another way to observe aliasing is to adjust the frequency slowly while watching the spectral display. If some of the harmonics are aliased, you will see the harmonics decreasing in frequency when they should be increasing or vice versa.

## 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



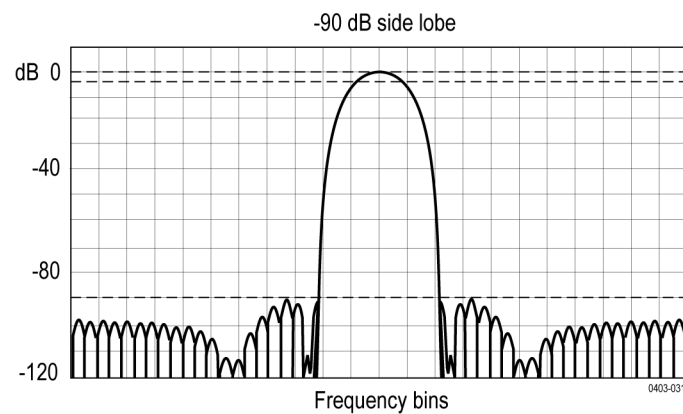
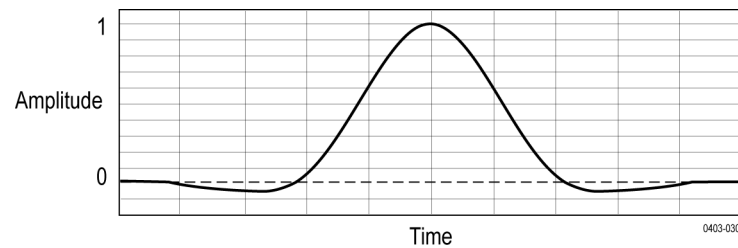
## FlatTop2 window

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.

---



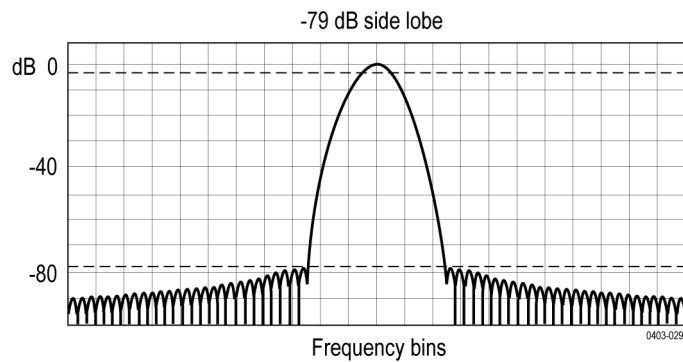
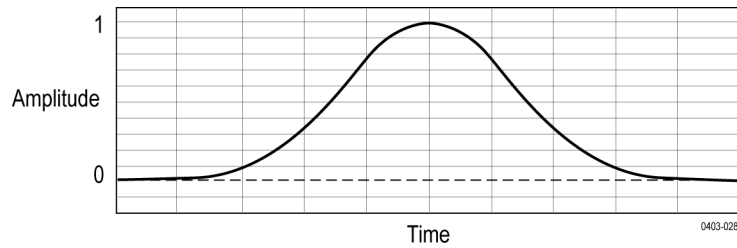
## Gaussian window

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.*

---

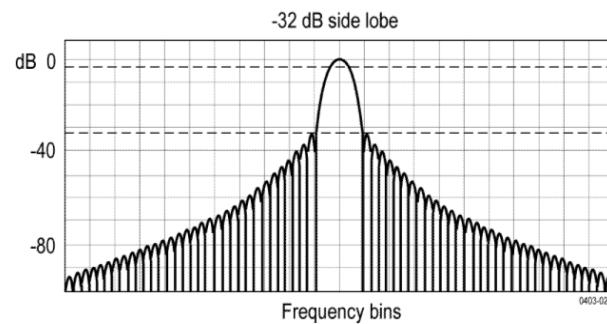
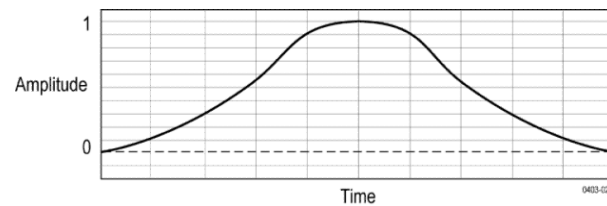


## Hanning FFT window

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.

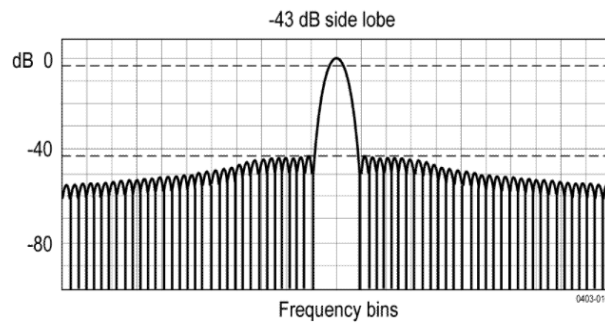
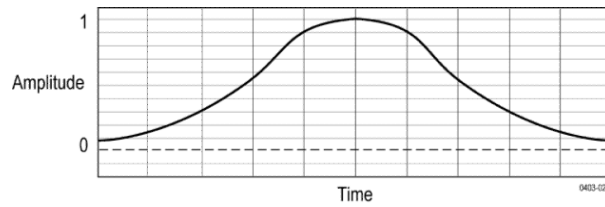
The Hanning window has the narrowest resolution bandwidth, but higher side lobes. Hanning has slightly poorer frequency resolution than Hamming. Hanning is best for measuring sine, periodic, and narrow-band random noise, and transients or bursts where the signal levels before and after the event are significantly different.

### Hanning window



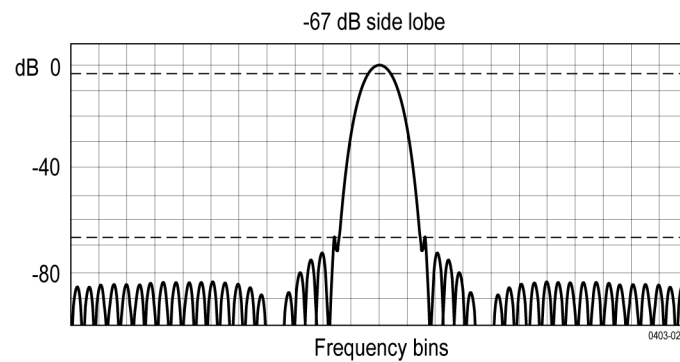
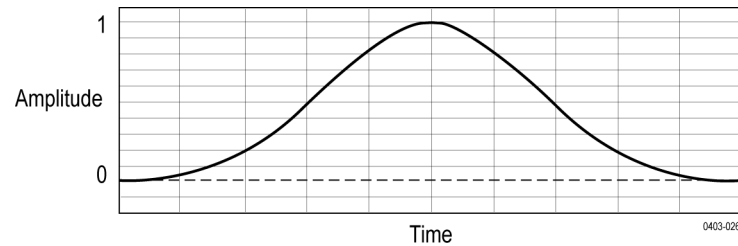
## Hamming window

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.



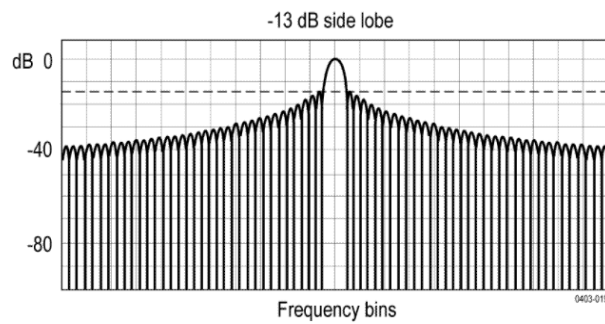
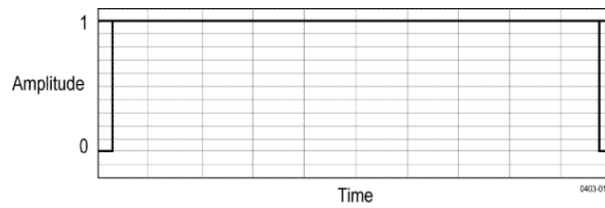
## Rectangular window

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

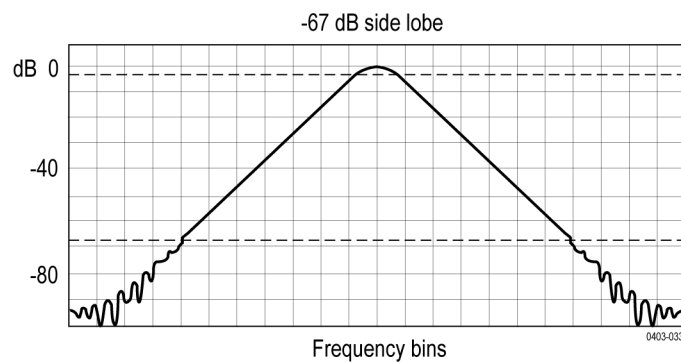
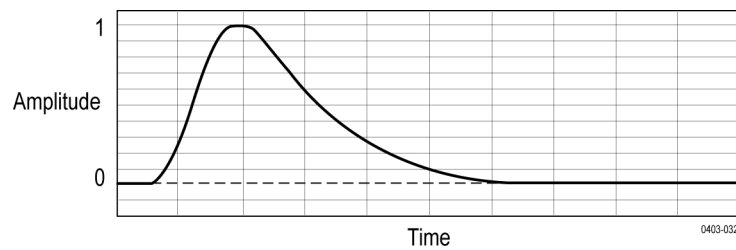
In the time domain, it is not a symmetrical bell shape as is the case with the other windows. Instead, it is exponential with a peak at the 20% position of the time domain gate. The frequency domain shape is triangular.

---

**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 ( $\mu$ ). 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.

$$Area = \int_{Start}^{End} Waveform(t) dt$$

Details of the integration algorithm are given later. [Integration algorithm](#) on page 610

### 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:

$$\int_A^B W(t)dt \text{ is approximated by } \int_A^B \hat{W}(t)dt \text{ where:}$$

0020-025

$W(t)$  is the sampled waveform

$\hat{W}(t)$  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,

$$\int_A^B (W(t))^2 dt \text{ is approximated by } \int_A^B (\hat{W}(t))^2 dt \text{ where:}$$

0020-027

$W(t)$  is the sampled waveform

$\hat{W}(t)$  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:

$$\int_A^B (\hat{W}(t))^2 dt = s \times \sum_{i=A}^{B-1} \frac{(W(i))^2 + W(i) \times W(i + 1) + (W(i + 1))^2}{3}$$

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.

$$Mean = \frac{\int_{Start}^{End} Waveform(t) dt}{(End - Start) \times SampleInterval}$$

0020-017

Details of the integration algorithm are given in [Integration algorithm](#) on page 610

**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.

$$\text{Negative Overshoot} = \frac{\text{Base} - \text{Min}}{\text{Amplitude}} \times 100 \%$$

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.

$$\text{Positive Overshoot} = \frac{\text{Max} - \text{Top}}{\text{Amplitude}} \times 100 \%$$

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.

$$RMS = \sqrt{\frac{(data1^2 + data2^2 \dots)}{(end - start)}}$$

**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**

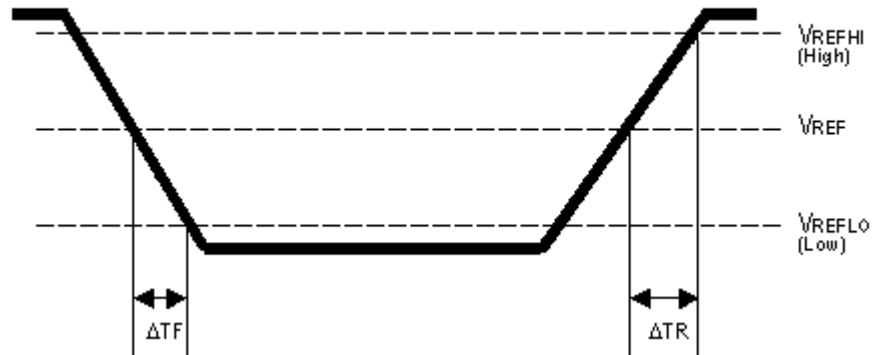
Delay is the time between a mid reference level edge on one source to a mid reference level edge on a second source. The direction of each edge can be configured by the user.

**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:

$$Falling\ Slew\ Rate = (VREF - VREFLO)/\Delta TF$$



### 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.)

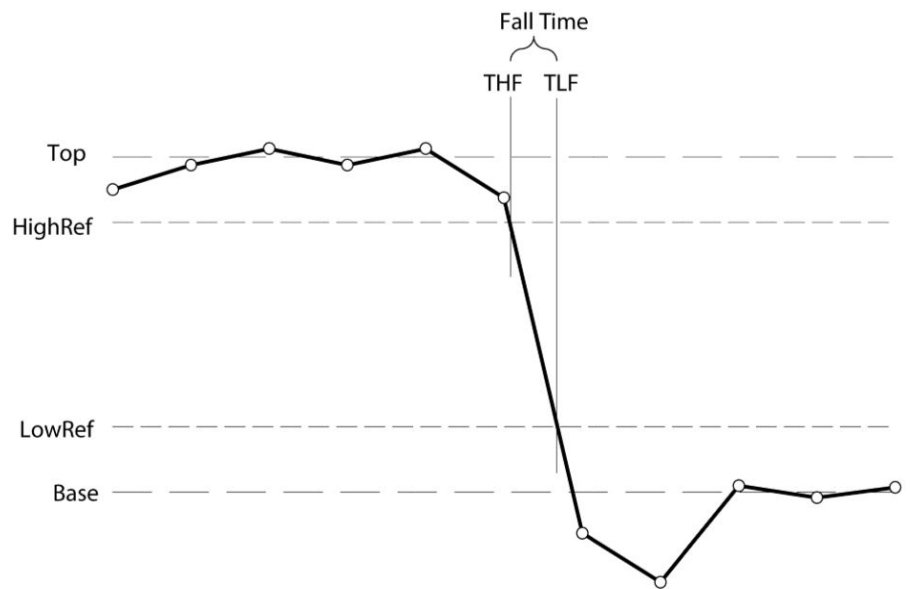


Figure 17: Fall Time

0403-004

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:

$NP_{Clock}$  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:

$NP_{Data}$  is the duration for N unit intervals.

$T_{Data}$  is the VRefMid crossing time in either direction.

If  $T_{n+N_{Data}}$  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.

$$\text{PositiveDutyCycle} = \frac{\text{PositiveWidth}}{\text{Period}} \times 100\% \quad 000421$$

**Positive Pulse Width measurement algorithm**

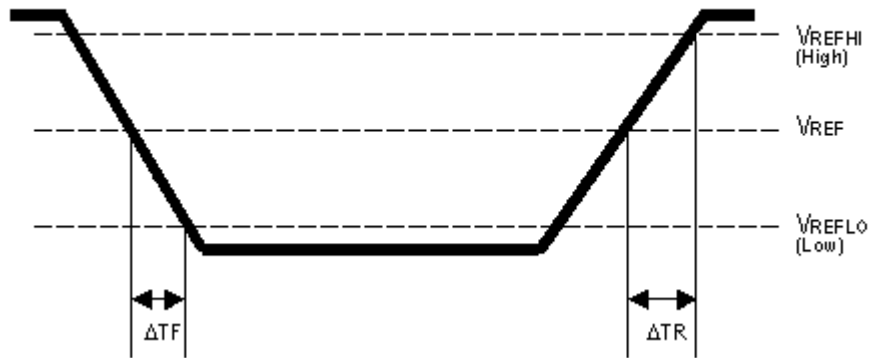
Positive Pulse Width is the time the signal remains above the mid reference level. It is the distance from a rising edge to the next falling edge.

**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:

$$(V_{REFHI} - V_{REF}) / \Delta TR$$

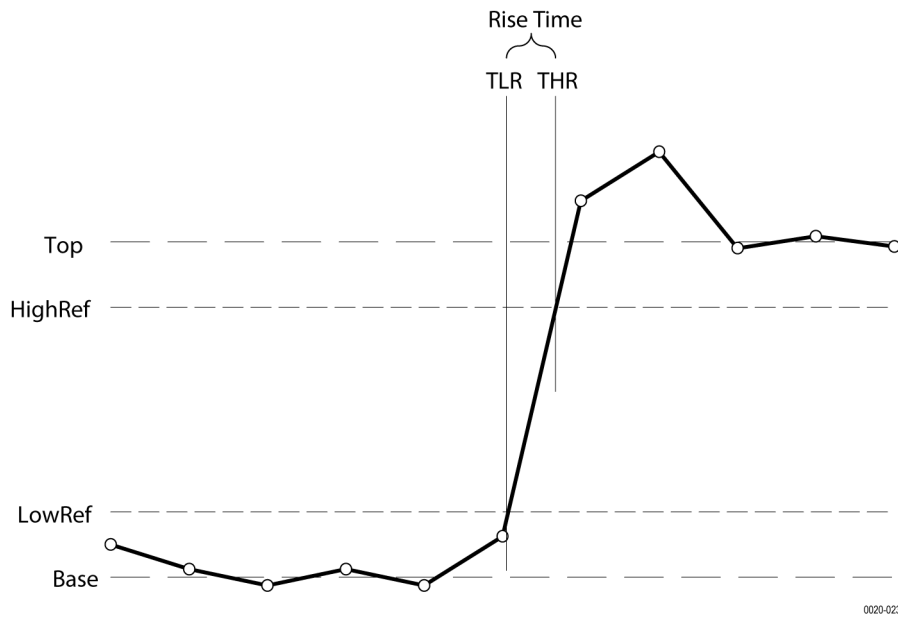


**Rise Time measurement algorithm**

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:

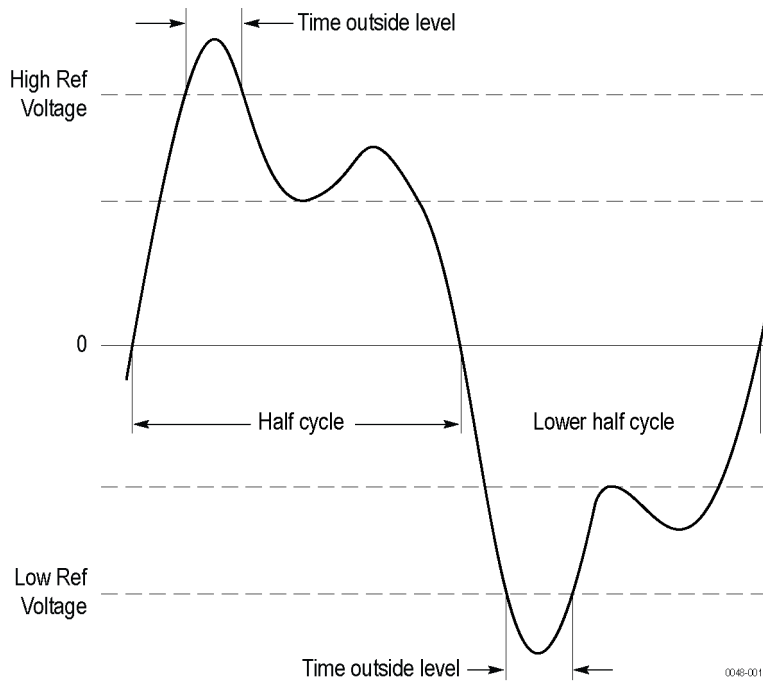
$T_{Skew}$  is the timing skew.

$T_{Main}$  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

<b>AC Common Mode</b>	<p>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:</p> $\text{Peak-to-Peak}(\text{High Pass Filter}((\text{Source1} + \text{Source2})/2))$
<b>Bit Amplitude measurement algorithm</b>	<p>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).</p>
<b>Bit High measurement algorithm</b>	<p>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).</p>
<b>Bit Low measurement algorithm</b>	<p>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).</p>
<b>DC Common Mode measurement algorithm</b>	<p>DC Common Mode is the arithmetic mean of the common mode of two sources. This measurement is made across the entire record.</p>
<b>Differential Crossover measurement algorithm</b>	<p>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.</p>

**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:

$$TIE_k^{Data} = T_k^{Data} - T'_k^{Data}$$

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.



---

<b>T/nT Ratio measurement algorithm</b>	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.
<b>DCD</b>	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.
<b>DDJ</b>	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.
<b>DJ</b>	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.
<b>Dual Dirac deterministic jitter</b>	Dual Dirac Deterministic Jitter ( $DJ-\delta\delta$ ) 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 $DJ-\delta\delta$ value is determined for each acquisition, by means of RJ-DJ separation analysis.
<b>F/2 measurement algorithm</b>	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.

<b>F/4 measurement algorithm</b>	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.
<b>F/8 measurement algorithm</b>	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.
<b>J2</b>	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.
<b>J9</b>	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.
<b>Jitter Summary measurement</b>	<p>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.</p>
<b>NPJ</b>	<p>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.</p> <p>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.</p>
<b>Phase noise</b>	<p>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.</p> <p>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.</p> <p>A Phase Noise measurement is required in order to enable the Phase Noise plot.</p>

- 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.

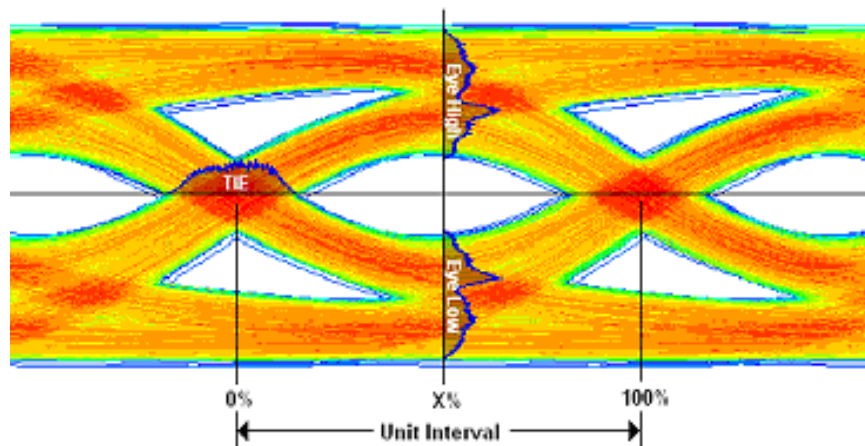
## Eye measurement algorithms

### 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.

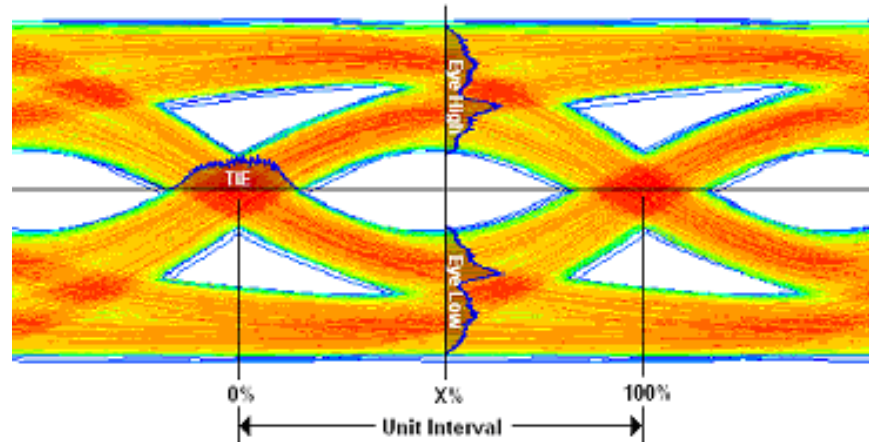



---

**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:

$$\text{Q-factor} = [\text{mean}(\text{EyeHigh}) - \text{mean}(\text{EyeLow})] / [\text{stddev}(\text{EyeHigh}) + \text{stddev}(\text{EyeLow})]$$

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 measurements: Input Analysis algorithms

Power measurements require option 5-PWR or 6-PWR.

## 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 ( $\theta$ ) 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:

$$I_{RMS} = \sqrt{\frac{1}{N} \sum_{n=0}^{N-1} i^2(n)}$$

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:

$$ApPwr = V_{RMS} \times I_{RMS}$$

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<sub>RMS</sub>* is the Root Mean Square of the current.

- **Phase Angle ( $\sigma$ ):** 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:

$$\phi = \cos^{-1} \frac{TrPwr}{ApPwr}$$

Where,  $\sigma$  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:

Re[k] is the Real component of kth harmonic.

Im[k] is the Imaginary component of kth harmonic.

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 discrete set of acquire time samples.

- **Harmonics  $F_k$**  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  $k$ th 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  $k^{\text{th}}$  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:

$$RMS = \sqrt{F1^2 + F2^2 + \dots + FOH^2}$$

$F1$  is the fundamental harmonics or line harmonics.

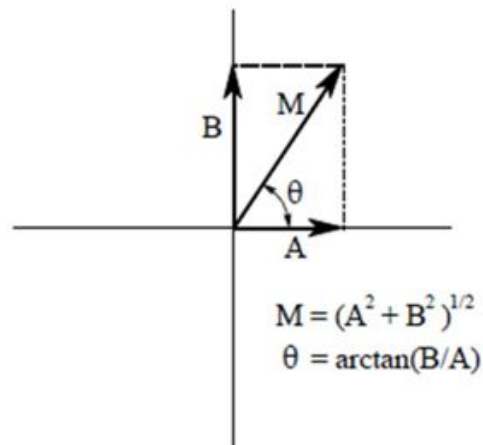
- **Total Harmonic distortion (THD-F):** It is measured as the ratio to the RMS value of the fundamental component of the source waveform. Reported as a percentage and calculated using the following equation:

$$THD - F = \sqrt{\frac{RMS^2 - F^2[k]}{F1}} \times 100\%$$

- **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:  $\sigma$ . 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] -  $\pi$

If, Rel[k] < 0 & Im[k] > 0 Phase[k] = Phase[k] +  $\pi$

- 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.

$$POHC(M) = \sqrt{\sum_{k=21,23}^{OH} F[k]^2}$$

Where:

M is the measured value.

POHC(S) and POHC(L) is the (S) is pass and fail status based on (L) limits.

### **Input Capacitance algorithm**

Input capacitance measures the input capacitance of a DUT using the input voltage and current signals. Supports annotation where you can navigate between Previous and Next regions from the Results badge.

The equation for input capacitance is  $c = q/v$ . where:

- c - capacitance in farads
- q - accumulated charge, which is the integration of the current waveform
- v - peak-peak voltage

### **Inrush Current algorithm**

Inrush Current measures the peak value of the inrush current of the DUT. It is a single source current measurement. Supports annotation where you can navigate between Previous and Next regions from the Results badge.

## Power measurements: Amplitude Analysis algorithms

Power measurements require option 5-PWR or 6-PWR.

### 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.

$$Cycle\ Amplitude = Top - Base$$

## Power measurements: Timing Analysis algorithms

Power measurements require option 5-PWR or 6-PWR.

### 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

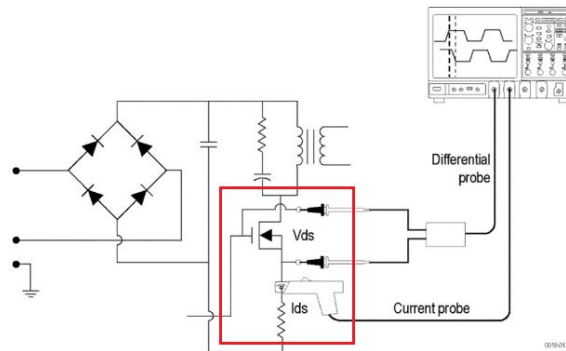
Positive Pulse Width is the time the signal remains above the mid reference level. It is the distance from a rising edge to the next falling edge.

## Power measurements: Switching Analysis algorithms

Power measurements require option 5-PWR or 6-PWR.

### 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 conversion. MOSFIT plays an important role to meet the design of SMPS.

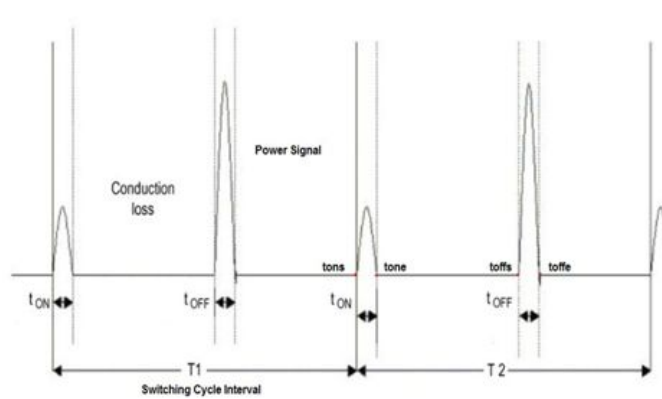
Regions of  $T_{on}$ ,  $T_{off}$ , and Conduction loss (Cond) with voltage source ( $V_{ds}$ ) and current source ( $I_{ds}$ ) are shown in Switching Loss  $T_{on}$ ,  $T_{off}$ , and Conduction loss regions.

All the switching losses are measured on Power signal, based on  $V_{ds}$  and  $I_{ds}$  transition.

$T_{on}$  Loss region: When  $V_{ds}$  starts rolling towards zero, the  $I_{ds}$  starts to roll upward.

$T_{off}$  Loss region: When  $V_{ds}$  starts rolling upwards, the  $I_{ds}$  start to roll towards zero.

Conduction Loss region: The region when the  $I_{ds}$  is high and  $V_{ds}$  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:

$$T_{oni} = f_{swi} \times \int_{T_{on-Start_i}}^{T_{on-Stop_i}} (V_{ds} \times I_{ds}) dt \text{ Watt}$$

$$T_{offi} = f_{swi} \times \int_{T_{off-Start_i}}^{T_{off-Stop_i}} (V_{ds} \times I_{ds}) dt \text{ Watt}$$

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.

$f_{swi}$  is the Switching frequency of the  $i^{th}$  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  $T_{off}$  region, in time unit  $V_{ds}$  - Voltage drain current, in Volts.

$I_{ds}$  is the drain current, in Amps.

■ **Energy Loss Computation**



Energy loss computation for  $T_{on}$  and  $T_{off}$  are calculated using the following equation:

$$T_{Eoni} = f_{swi} \times \int_{T_{on-Start_i}}^{T_{on-Stop_i}} (V_{ds} \times I_{ds}) dt \text{ Watt}$$

$$T_{Eoffi} = f_{swi} \times \int_{T_{off-Start_i}}^{T_{off-Stop_i}} (V_{ds} \times I_{ds}) dt \text{ Watt}$$

Where:

$T_{Eoni}$  is the  $i^{\text{th}}$  switching cycle turn on energy loss in joule.

$T_{Eoffi}$  is the  $i^{\text{th}}$  switching cycle turn off energy loss in joule.

#### ■ Computation of Conduction

Conduction is computed as  $R_{DS(on)}$  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:

$$Cond_i = f_{swi} \times \int_{T_{on-Stop_i}}^{T_{off-Start_i}} (R_{ds(on)} \times I_{ds}^2) dt \text{ Watt}$$

$$Cond_{Ei} = \int_{T_{on-Stop_i}}^{T_{off-Start_i}} (R_{ds(on)} \times I_{ds}^2) dt \text{ Joule}$$

Where:

$R_{DS(on)}$  is the Dynamic resistance, in  $\Omega$ .

$Cond_i$  is the Conduction Loss in Watt,  $Cond_{Ei}$  is the Conduction Energy Loss in Joule.

$Cond_i$  is the  $i^{\text{th}}$  switching cycle conduction loss in watt.  $Cond_{Ei}$  is the  $i^{\text{th}}$  switching cycle conduction energy in joules.

$f_{swi}$  is the  $i^{\text{th}}$  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:

$$Cond_i = f_{swi} \times \int_{T_{on-Stop_i}}^{T_{off-Start_i}} (V_{ce(SAT)} \times I_{ds}) dt \text{ watt}$$

$$Cond_{Ei} = \int_{T_{on-Stop_i}}^{T_{off-Start_i}} (V_{ce(SAT)} \times I_{ds}) dt \text{ joule}$$

Where,  $V_{CE(sat)}$  is the voltage in volt, which should be configure in application.

#### ■ Computation of Average Loss and Total Loss

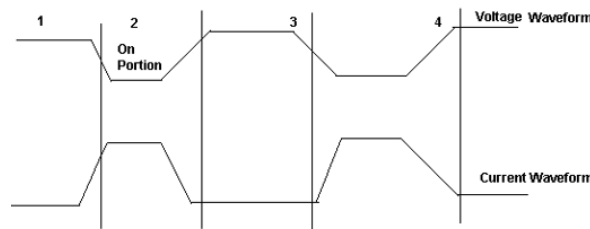
Average and Total loss are calculated using the following equation:

$$T_{onAvg} = \frac{1}{N_c} \sum_{i=1}^{N_c} T_{on_i}$$

$$T_{offAvg} = \frac{1}{N_c} \sum_{i=1}^{N_c} T_{off_i}$$

$$Cond_{Avg} = \frac{1}{N_c} \sum_{i=1}^{N_c} Cond_i$$

$$TotalLoss = T_{onAvg} + T_{offAvg} + Cond_{Avg}$$



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.

**$R_{DS(on)}$  algorithm** Dynamic resistance ( $R_{DS(on)}$ ) is the resistance offered by a switching device when it is in the ON condition. Power helps to monitor the dynamic resistance using a Time Trend plot. You can calculate  $R_{DS(on)}$  by using the below formula:

$$R_{DS(on)} = v/i = v(t)/i(t)$$

where,  $v/i$  is the ratio of voltage to current sample points. It is applicable for switching semiconductors.

## Power measurements: Magnetic Analysis algorithms

Power measurements require option 5-PWR or 6-PWR.

The Magnetic measurements include I vs.  $\int V$ , Inductance, Magnetic Property, and Magnetic Loss. When using I vs.  $\int V$  measurement, take care to check that the voltage 'V' does not have any DC components.

Use AC coupling on the input signal to avoid any DC shifts on the integral of the voltage waveform.

**Inductance algorithm** It measures the inductance (the integral of the voltage divided by the current) of a magnetic component during in-circuit operation. The application creates a single cycle by averaging multiple cycles of current and integrated voltage.

An electric circuit has electromotive force created by a change of current in the same circuit known as self-inductance or in a neighboring circuit as mutual-inductance. Inductance displays the Inductance plot. The unit of Inductance is Henry.

The Inductance provides a view of the core behavior while it is under operation.

**I vs (integral of) V algorithm**

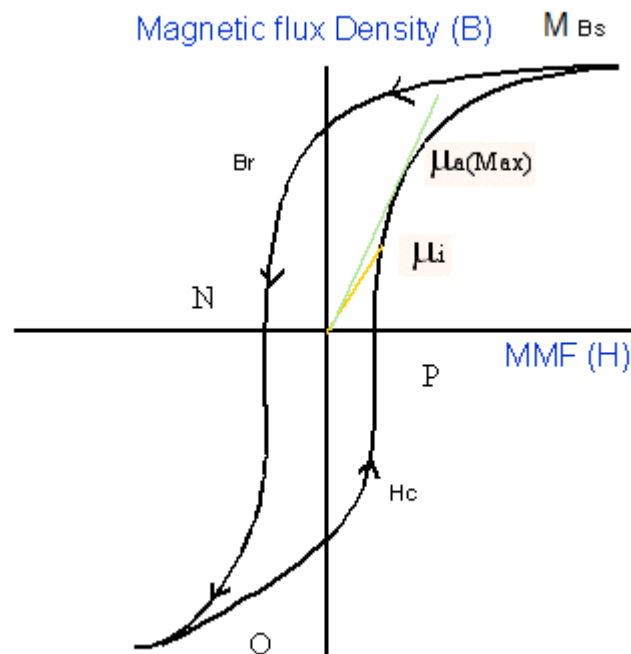
The XY plot of integral of voltage against current. The integral of the voltage is proportional to B. The integral of current is proportional to H.

**Magnetic Loss algorithm**

The average value of the product of the voltage times current through the inductor. This represents the total loss of the magnetic device and consists of resistive and eddy current losses during circuit operation.

**Magnetic Property algorithm**

The following diagram shows a plot of Hysteresis in a typical magnetic material (magnetic field strength (H) versus saturation flux density (B)).



**Figure 18: B-H curve**

$B_s$  is the Saturation Flux Density

$B_r$  is the Remanence Flux Density

$H_c$  is the Coercive Force ( $H_c$ )

$\mu_i$  is the Initial Permeability

$\mu_a$  is the Max Amplitude permeability

H is the magnetic field used to induce Magnetic Flux in the magnetic material

MMF is Magneto Motive Force, it is also known as Magnetic Field Strength

**NOTE.** The data waveform starts from the Max value of H, decreases, and then increases again (M-N-O-P).

**Magnetic Field Strength (H)**

The previous figure shows the hysteresis in a typical magnetic material. The magnetic field induces a magnetic flux in the DUT. The units of measurement are Ampere per meter in SI unit, and Oersted in CGS unit.

### Saturation Flux Density ( $B_s$ )

The Saturation Flux Density represents maximum magnetic flux density that can be induced in the magnetic material regardless of the magnitude of the externally applied field H. This is represented in the B-H curve where B value is considered, when H is maximum.

$$B_s = \text{Max}(B_k)$$

The Magnetic Field Intensity H is also calculated on the maximum flux density cycle  $B_k$ .

$$B_s = \text{Max}(B)$$

Index I where H is maximum

$$I = \text{Index of}(\text{Max}(H))$$

$$B_s = B(I) \text{ (2)}$$

### Remanence ( $B_r$ ):

Remanence is the Induced magnetic flux density that remains in the material after the externally applied magnetic field (H) is returned to zero during the generation of the hysteresis loop. This represents max value of B for all values of zero value of H in the B waveform.

Find the index at zero value of H on the H waveform and calculate the maximum value of B from these indices.

Let 'q' be the index at zero value of H on H waveform. Let the q1 and q2 be the indices of the waveform. Calculate the value of B at the Indices q1 and q2 on the  $K^{\text{th}}$  cycle. The maximum magnitude value of B is the remanence Flux density.

### Coercive Force( $H_c$ )

Coercivity is the value of H found at the intersection of the H-axis with the hysteresis loop. This represents the external field required to cause the induced flux density (B) to reach zero during the measurement cycle of a hysteresis loop.  $H_c$  is symmetrical with the positive and negative axis.

Coercivity  $H_c$  is calculated on the cycle where the maximum flux density occurs in the entire acquired waveform.

Finding the index at zero B value on the B waveform: Let 'q' be the index at zero value of B on B waveform. Let q1 and q2 be the indices on the B waveform where B is zero.

Coercivity is the maximum magnitude of the H data at the indices of q1 and q2 is the coercivity.

### Permeability

The ratio of B and H calculated on the  $B_k$  cycle. Select the points on the B-H plot using the cursor and calculate the slope of the B-H curve using the data selected between the cursor. You can choose the portion of the plot using the cursor to obtain the results.

In the B-H curve plot there is a provision to select the points using the waveform cursors 1 and 2. This is used to calculate the slope of B-H curve using the data between the cursors 1 and 2. The computed slope as explained below displays the scalar value of the permeability ( $\mu$ ) in the results badge and table.

In slope calculation, there are N points between the cursors:

Find

$$H_{av} = (H1+H2+\dots Hn)/N$$

$$B_{av} = (B1+B2+\dots Bn)/N$$

$$H_{normi} = Hi-H_{av}, i=1..N$$

$$B_{normi} = Bi-B_{av}, i=1..N$$

$$B/H = \text{SUM}(H_{normi}1*B_{normi}1+H_{normi}2*B_{normi}2+\dots+H_{normi}N*B_{normi}N) / \text{SUM}(H_{normi}1*H_{normi}1+H_{normi}2*H_{normi}2+\dots+H_{normi}N*H_{normi}N)$$

Where,

B/H is the Permeability ( $\mu$ )

## Power measurements: Output Analysis algorithms

Power measurements require option 5-PWR or 6-PWR.

### Line Ripple algorithm

Line Ripple measures the voltage of a power supply which has been derived from an alternating current (AC) source, displaying the results as a peak-to-peak value and RMS value. In Line Ripple measurements, the time base is set to display three cycles of 50 Hz or 60 Hz in the input waveform.

Depending upon the coupling type you select, the application sets the required offset and adjusts the vertical scale to the appropriate sensitivity.

### Switching Ripple algorithm

Switching Ripple measures the voltage ripple at operating the specified power supply switching frequency, up to 1 MHz.

**Efficiency algorithm**

Efficiency measures the ratio of output power to input power for a power conversion circuit. The measurement supports three outputs, the maximum and efficiency is computed at each output. If there are more than one output, then the total efficiency value will be computed. You can configure AC/DC type at input and output sides.

---

**NOTE.** *This measurement uses a minimum of four sources and a maximum of eight sources. Two sources are used to measure the input voltage and current to the power supply. Two to six sources are used to measure the power supply output voltage and current.*

---

Math is used to compute Input Power waveform using  $V * I$  equation. Computation happens over an exact integer number of cycles to get an accurate result, that is, from the first zero crossing to the last zero crossing.

Math calculates the Input Power, using

$$\text{Math1} = V(t)_{in} * I(t)_{in}$$

Similarly another math calculates the Output Power using  $V * I$  at the output side

$$\text{Math2} = V(t)_{o1} * I(t)_{o1}$$

Power Efficiency = Output Power / Input Power

$$\text{Efficiency in \%} = \frac{\sum(\text{Math1})}{\sum(\text{Math2})} * 100$$

---

**NOTE.**

- *If more cycles are captured, then the power solution will automatically consider all the integral cycles, starting from first (mid) crossing to the last valid (mid) crossing.*
  - *For multiple outputs, the total efficiency is computed as the ratio of input power to the sum of output(s) power. The total efficiency is expected to be within 100%, since all the outputs are from a same input power. If the efficiency is greater than 100 %, then the application displays a warning message.*
-



- Turn On Time** **Turn On Time** is the time taken to get the steady state output voltage of the power supply after the input voltage is applied.
- The **Maximum Voltage** value should be greater than the **Input Trigger** level.
  - The **Maximum Voltage** level value sets the waveform vertical settings to approximately one-fourth of the specified level.
  - The **Maximum Time** field sets the horizontal units so as to display the entire time in which the output is supposed to reach zero.
  - The **Input Trigger** level is set based on the 50 % level of the input channel.
  - **Turn Off Time** sets the trigger slope to rising edge.
- Turn Off Time** **Turn Off Time** is the time taken to get the output voltage to low level (close to zero) after the input voltage is removed.
- The **Maximum Voltage** value should be greater than the **Input Trigger** level.
  - The **Maximum Voltage** level value sets the waveform vertical settings to approximately one-fourth of the specified level.
  - The **Maximum Time** field sets the horizontal units so as to display the entire time in which the output is supposed to reach zero.
  - The **Input Trigger** level is set based on the 50 % level of the input channel.
  - **Turn Off Time** sets the trigger slope to rising edge.

## Power measurements: Frequency Response Analysis algorithms

Power measurements require option 5-PWR or 6-PWR.

### Control loop response measurement algorithm

Control Loop Response computes and plots gain as  $20 \log (V_{\text{out}}/V_{\text{in}})$  and phase difference between  $V_{\text{in}}$  and  $V_{\text{out}}$  at each frequency within the swept band. The resulting plot is commonly referred to as a Bode Plot.

Points per decade is 10 by default, maximum is 100.

Start frequency is 100 Hz

Stop frequency is 10 MHz

$$\text{Number of frequency points} = \text{ppd} * (6 - 2)$$

where:

- 6 is  $10^6 = 10 \text{ MHz}$
- 2 is  $10^2 = 100 \text{ Hz}$

The gain and phase margin are two metrics to tell the stability of the system.

$$Bode = 20 * LOG10\left(\frac{V_{out}}{V_{in}}\right)$$

Gain for

where:

- $V_{out}$  is the amplitude of the input signal
- $V_{in}$  is the input signal

Phase for Bode is the time shift between the output and input signals.

**Power supply rejection ratio measurement algorithm**

Power Supply Rejection Ratio (PSRR) measures both the modulated input and output AC voltage levels and then computes the rejection ratio as  $20 \log (V_{in}/V_{out})$  at each frequency within the swept band.

## DDR Amplitude Measurement algorithms

DDR measurements require option 6-DBDDR3 (6 Series MSO instruments only).

- AOS (DDR)** AOS is the total area of the signal above specified reference level.
- AOS Per tCK (DDR)** AOS Per tCK is the total area of the signal that crosses the specified reference level, calculated over consecutive periods. It is applicable to clock and address/command waveforms.
- AOS Per UI (DDR)** AOS Per UI is the total area of the signal that crosses specified reference level, calculated over consecutive unit intervals. It is applicable to data and data strobe waveforms.
- AUS (DDR)** AUS is the total area of the signal below specified reference level.

<b>AUS Per tCK (DDR)</b>	AUS Per tCK is the total area of the signal that crosses the specified reference level, calculated over consecutive periods. It is applicable to clock and address/command waveforms.
<b>AUS Per UI (DDR)</b>	AUS Per UI is the total area of the signal that crosses specified reference level, calculated over consecutive unit intervals. It is applicable to data and data strobe waveforms.
<b>Vix(ac) (DDR)</b>	Vix(ac) is the differential input cross-point voltage measured from the actual crossover voltage and its complement signal to a specified reference voltage. This is measured on the single ended signal.

## DDR Time Measurement algorithms

DDR measurements require option 6-DBDDR3 (6 Series MSO instruments only).

<b>tRPRE (DDR)</b>	tRPRE is the width of the Read burst' preamble. This is measured from the exit of tristate to the first driving edge of the differential strobe.
<b>tWPRE (DDR)</b>	tWPRE is the width of Write burst preamble. It is measured from the exit of tristate to the first driving edge of the differential strobe.
<b>tPST (DDR)</b>	tPST is the width of Read or Write burst postamble. It is measured from the last falling edge crossing mid reference level to the start of an undriven state (judged by a rising trend as per JEDEC specification).
<b>Hold Diff (DDR)</b>	Hold Diff is the elapsed time between the designated edge of the single ended waveform and the designated edge of a differential waveform. The closest single-ended waveform edge to the differential waveform edge that falls within the range limits is used for the measurement.

**Setup Diff (DDR)** Setup Diff is the elapsed time between the designated edge of a single ended waveform and when the differential waveform crosses its own voltage reference level. The closest single ended waveform edge to the differential waveform edge that falls within the range limits is used for the measurement.

**tCK(avg) (DDR)** tCK(avg) is the average clock period across a sliding 200-cycle window.

$$tCK(avg) = \left[ \sum_{j=1}^N tCK_j \right] / N$$

Where,

N=200 (configurable)

**tCH(avg) (DDR)** tCH(avg) is the average high pulse width calculated across a sliding 200 cycle window of consecutive high pulses.

$$tCH(avg) = \left[ \sum_{j=1}^N tCH_j \right] / (N * tCK(avg))$$

Where,

N=200 (configurable)

**tCL(avg) (DDR)** tCL(avg) is the average low pulse width calculated across a sliding 200 cycle window of consecutive low pulses.

$$tCL(avg) = \left[ \sum_{j=1}^N tCL_j \right] / (N * tCK(avg))$$

Where,

N=200 (configurable)

**tCH(abs) (DDR)** tCH(abs) is the high pulse width of the differential clock signal. It is the time duration, the waveform remains above the mid reference voltage level.

$$tCH(abs) = tCH / tCK(avg)$$

**tCL(abs) (DDR)** tCL(abs) is the low pulse width of the differential clock signal. It is the amount of time the waveform remains below the mid reference voltage level.

$$tCL(abs) = tCL / tCK(avg)$$

**tJIT(duty) (DDR)** tJIT(duty) is the largest elapsed time between tCH from tCH(avg) or tCL from tCL(avg) for a 200-cycle window.

$$tJIT(duty) = \text{Min/max of } \{tJIT(CH), tJIT(CL)\}$$

Where,

$$tJIT(CH) = \{tCH_i - tCH(avg)\}$$

$$tJIT(CL) = \{tCL_i - tCL(avg)\}$$

i =1 to N

N=200 (configurable)

**tJIT(per) (DDR)** tJIT(per) is the largest elapsed time between the tCK from tCK(avg) for a 200-cycle sliding window.

$$tJIT(per) = \text{Min/Max of } \{tCK_i - tCK_{(avg)}\}$$

Where,

i =1 to N

N=200 (configurable)

**tJIT(cc) (DDR)** tJIT(cc) is the absolute difference in clock period between two consecutive clock cycles.

$$tJIT(cc) = \text{Max of } |tCK_{i+1} - tCK_i|$$

**tERR(n) (DDR)** tERR(n) is the cumulative error across multiple consecutive cycles from tCK(avg). It measures time difference between the sum of clock period for a 200-cycle window to n times tCK(avg).

$$tERR(nper) = \left( \sum_{j=1}^{i+n-1} tCK_j \right) - n * tCK(avg)$$

Where,

n = 2 for tERR(2 per)

n = 3 for tERR(3 per) and so on.

**tERR(m-n) (DDR)** tERR(m-n)  
tERR(m-n) is the cumulative error across multiple consecutive predefined cycles from tCK(avg). This is measured similar to tERR(n per).

$$tERR(nper) = \left( \sum_{j=1}^{i+n-1} tCK_j \right) - n * tCK(avg)$$

Where,

$6 \leq n \leq 10$  for tERR (6-10 per)

$11 \leq n \leq 50$  for tERR (11-50 per)

**tDQSCK (DDR)** tDQSCK is the strobe output access time from differential clock. tDQSCK is measured between the rising edge of clock before or after the differential strobe Read preamble time. The edge locations are determined by the mid-reference voltage levels.

- tCKSRE** tCKSRE is the valid clock cycles required after Self Refresh Entry (SRE) command. Changing the input clock frequency or the supply voltage is permissible only after tCKSRE time when the SRE command is registered.
- tCKSRX** tCKSRX is the valid clock cycles required before the Self Refresh Exit (SRX) command. Changing the input clock frequency or the supply voltage is permissible provided the new clock frequency or supply voltage is stable for the tCKSRX time prior to SRX command.
- tCMD-CMD** tCMD-CMD is the elapsed time between two logic states.



# References

## Power badge error and warning messages (optional)

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	The 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 are 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 are 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 are set in the measurement configuration menu Reference Levels panel.

Error Message	Cause	Suggestion
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 are set in the measurement configuration menu Reference Levels panel.
Invert probing points	Polarity mismatch between the probe to test points on the DUT	Check and inverse the probe polarity to match with DUT test points.
Too few edges	The oscilloscope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Use <b>Power Autose</b> t to optimize the oscilloscope settings for power measurements.
Insufficient sampling rate	Not enough sampling rate used to capture signal	Increase the sample rate proportional to input signal frequency. Use <b>Power Autose</b> t to optimize the oscilloscope settings for power measurements.
Too few cycles	Not enough sampling rate used to capture signal	Increase the sample rate proportional to input signal frequency. Use <b>Power Autose</b> t to optimize the oscilloscope settings for power measurements.
Not enough data	The oscilloscope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Use <b>Power Autose</b> t to optimize the oscilloscope settings for power measurements.
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.

Error Message	Cause	Suggestion
Insufficient horizontal resolution	Not enough sampling rate used to capture signal	Increase the sample rate proportional to input signal frequency. Use <b>Power Autose</b> to optimize the oscilloscope settings for power measurements.
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 are set in the measurement configuration menu Reference Levels panel.
Not enough data	The oscilloscope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Use <b>Power Autose</b> to optimize the oscilloscope settings for power measurements.
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 are set in the measurement configuration menu Reference Levels panel.
Too few cycles	The oscilloscope has acquired less than one complete SW cycle.	Increase time base to capture more than one complete cycle. Use <b>Power Autose</b> to optimize the oscilloscope settings for power measurements.
Not enough data	Table results will not show values.	Check input waveforms and configuration. Recommended to run Power Autose and then observe the measurement results.
Too few edges	No of edges found in the gated region is less than the number of edges required for the algorithm for calculation.	Increase the record length.
Pos clipping	Vertical scale is not set properly.	Use <b>Power Autose</b> to set the vertical scale automatically.
Neg clipping	Vertical scale is not set properly.	Use <b>Power Autose</b> to set the vertical scale automatically.
Invalid range Indices	No data in the gated region.	Make sure that the input waveforms are valid.
Empty Input	Input waveform has no data.	Make sure to set all the valid inputs.
Input all DC	Input signal is pure DC.	Change the input signal to AC or add some AC component to DC signal.
No data in range	No data in the gated region.	Make sure that the input waveforms are valid.
Unable to connect to generator	Unable to connect to the generator at the specified IP address.	Check the IP address, verify that the generator is powered on, and try to run the measurement again.
No AFG option	AFG option is disabled.	An AFG license is needed. See <a href="#">Advanced Power Analysis option</a> on page 21.

Error Message	Cause	Suggestion
No Results	Results are not available	An Internal error has occurred. Results are not available. Check the connections and try again.
Ampl set error	Measurement Runtime Error.	The generator amplitude and impedance values do not match. Restart the measurement.
V/H scale error	Unable to perform vertical or horizontal scaling	Restart the measurement.
Runtime error	Measurement Runtime Error.	Windows OS: Exit the TekScope application and relaunch again. Standard instrument (not Windows OS): Restart the oscilloscope.
Incompatible meas	When any other measurement is added in addition to FRA measurement.	Delete all the measurements other than FRA measurement. There should be only one measurement at a time.

### 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 are 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 are 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 are 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.

Warning Message	Cause	Suggestion
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 are 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.
No data in range	No data in the gated region.	Make sure that the input waveforms are valid.
Input > 80% DC	Input signal has higher DC component.	Change the input signal to AC or add more AC component to the DC signal.
Trigger Level is greater than maximum voltage	Trigger level set is greater than the maximum voltage.	Set the maximum voltage higher than the trigger level.
Output Voltage Level reached before input trigger	Output is turned on before the input or there is a spike in the output voltage.	Make sure that the input and output voltage are not interchanged, output is dependent on the input, and run the measurement again.
No PM value	No zero dB crossing on the Gain curve.	Check the test setup connections and try again.
No GM value	No zero degree crossing on the Phase curve.	Check the test setup connections and try again.

**Information messages  
displayed on the Power  
measurement badges**

Information messages	Description
Actual Input Level used in calculation is n V.	In Turn On Time and Turn Off Time measurements, if the input level is not equal to the maximum of input waveform (n), the maximum of the input waveform is used for computation and the same is displayed as the information message.
Actual Output x Level used in calculation is n V.	In Turn On Time and Turn Off Time measurements, if the output x level is not equal to the maximum output x waveform (x), the maximum of the output x waveform is used for computation and the same is displayed as the information message.

# Index

250 Kohm termination,  
5-AFG, SUP5-AFG option, 17  
5-SEC enhanced instrument security option, 18  
5-WIN, SUP5-WIN option, 16  
6-SEC enhanced instrument security option, 18  
6-WIN, SUP6-WIN option, 16

## A

A B Sequence trigger,  
A event trigger menu,  
A knob, 44  
about the instrument,  
AC common mode,  
AC line voltage, 574  
AC RMS,  
accessories. standard, 5  
Accessory pouch, 5  
ACQ acquisition status LED, 54  
Acquired waveform, 570  
acquiring a signal, 99  
Acquisition  
    input channels and digitizers, 563  
    sampling, 563  
Acquisition concepts, 563  
Acquisition controls, 43  
Acquisition menu  
    Clear, 203  
    Run/Stop, 203  
    Save on Trigger, 203  
    Single/Seq, 203  
acquisition menu, open, 104  
acquisition mode, 203  
acquisition modes, 565  
Active probes, 8, 569  
add  
    a bus waveform to the display, 272  
    a channel to the display, 105  
    a label to a waveform, 134  
    a measurement badge, 112  
    a measurement plot, 117  
    a note to the screen, 134

    a search badge, 127  
    a vertex (visual trigger), 155  
    DDR measurement plot,  
    DDR measurements,  
    filter expression to math equation,  
    function to math equation,  
    measurement statistical readouts to badge, 64  
    signal to the screen, 105  
    statistic readouts to measurement badge, 228  
    visual trigger area to screen, 154  
    waveform to the screen, 105  
add a plot, 306  
Add Filter (math equation editor),  
Add Filter (math waveform),  
Add Measurements menu, 210  
Add New  
    Bus waveform button, 59  
    Math waveform button, 59  
    Ref waveform button, 59  
add results table menu, 307  
Add Variable (math waveform),  
Add Variable dialog,  
adding math waveforms,  
advanced jitter analysis option, 20  
advanced math waveform, 397  
advanced triggering, 145  
AFG option, 59  
AFG Out (rear panel), 56  
AFG Settings, 388  
AFG Sync out,  
aliasing, 599  
Aliasing  
    recognizing, 599  
Amplitude Analysis  
    cycle maximum,  
    cycle minimum,  
Amplitude Measurement algorithms  
    amplitude,  
    cycle base,  
    Cycle Top,  
amplitude measurements  
    AC RMS,

- Amplitude,
- Area,
- Base,
- Maximum,
- Mean,
- Minimum,
- Negative Overshoot,
- Peak-To-Peak,
- Positive Overshoot,
- RMS,
- Top,

Amplitude Measurements panel,

analog channels, 379

annotations,

AOS Per tCK measurement (DDR),

AOS Per UI measurement (DDR),

Arbitrary function generator (option 5-AFG, SUP5-AFG), 17

arbitrary/function generator, 388

Area,

Area configuration menu (Visual Trigger), 555

ARINC429,

Audio bus trigger settings,

audio serial bus menu,

AUS Per tCK measurement (DDR),

AUS per UI measurement (DDR),

auto serial bus menu,

auto-dim screen,

automatic probe compensation (TPP0500B, TPP1000), 83

Autoset, 100

Autoset button, 43

Autoset, disable/enable,

AUX Out,

AUX Out (rear panel), 56

Aux Trig trigger input, 54

AUX trigger, 154

Average

- acquisition mode, 203

average trace (spectrum view),

avoid pinching when rotate handle, 30

## B

B knob, 44

B trigger event menu,

B-H curve plot, 123

B-H curve plot menu,

backlight,

badge groups

- badge group caveats, 71
- badge group characteristics, 71
- how to create a group, 71

badge types, 63

badges, 63, 121, 175

Bandwidth options, 11

bar graph, 125

bar graph plot menu,

Base,

Bathtub Plot menu,

binary virtual keypad, 554

Blackman-Harris FFT window,

Blackman-Harris window

- defined, 600

browse save as location menu,

Burst Width,

Bus badge, 272

Bus button, 43

Bus Decode Results table,

Bus Decode Table menu,

bus inputs, parallel,

bus menu, parallel,

bus search,

bus setup, 139

bus setup menus, 272

Bus trigger

- defined,

bus waveforms, 111

Butterworth filter,

## C

cable lock, 56

calculate reference levels,

Calibration certificate, 5

CAN bus

- .dbc symbolic search config menu,

CAN bus trigger settings,

CAN dbc, 279

CAN serial bus menu,



- capturing screen shots,
  - change area height (visual trigger), 158
  - change area source (visual trigger), 158
  - change area width (visual trigger), 158
  - change area's center (visual trigger), 158
  - change display settings, 129
  - change measurement settings, 115
  - change password (Opt 5-SEC),
  - change password (Opt 6-SEC),
  - change security password (Opt 5-SEC),
  - change security password (Opt 6-SEC),
  - change waveform colors,
  - changing channel handle font, color, size, 443
  - changing note font, color, size, 443
  - changing waveform handle font, color, size, 443
  - channel badge, 63
  - channel badge (spectrum view), 175
  - channel badge menu, 379
  - Channel buttons (front panel), 43
  - channel settings menu, 379
  - Channel Settings menu
    - Other drawer,
  - Clear, 105
  - clear acquired waveform data, 203
  - Clear button, 43
  - Clipping, 570
  - clipping message, 63
  - Clock Edge,
  - clock format (12/24 hr), how to set, 79
  - Clock Offset,
  - Clock Recovery panel,
  - Clock Recovery- Advanced Settings,
  - common touchscreen UI tasks, 75
  - compensate passive probes, 85
  - compensate probes,
  - compensate TPP0500B, TPP1000 probes, 83
  - configuration menus, 72
  - configure a measurement, 115
  - configure cursors (spectrum view),
  - Configure panel (DDR),
  - configure profile,
  - connect a monitor, 96
  - connect digital signals, 137
  - connect lock cable, 39
  - connect to a network, 86
  - connecting probes, 40
  - Constant Clock,
  - contact Tektronix, 3
  - Control Loop Response,
  - Control Loop Response plot, 126
  - Control Loop Response plot menu,
  - Control windows
    - vertical acquisition, 571
  - copy files,
  - correct handle rotate, 30
  - Coupling, 569
  - create visual trigger, 154
  - Creating
    - create math waveforms, 591
  - Current probes, 8
  - Cursor measurements, 596
  - cursor readout (spectrum view),
  - cursor settings, 390
  - cursors
    - H Bar, 390
    - linked, 390
    - split, 390
    - V Bar, 390
    - V&H Bar, 390
    - waveform, 390
  - Cursors button, 43
  - Cursors button (touchscreen), 57
  - cursors menu, 130
  - cursors menu (spectrum view),
  - custom waveform colors,
  - customizing measurements, 187
- ## D
- Damping,
  - Data Rate,
  - date, 391
  - dbc (CAN bus),
  - DC offset, 597
  - DCD, 623
  - DDJ, 623
  - DDR

- add plot,
- amplitude measurements,
- AOS,
- AOS Per tCK,
- AOS Per tCK measurement,
- AOS Per UI,
- AOS Per UI measurement,
- AUS,
- AUS Per tCK,
- AUS Per tCK measurement,
- AUS Per UI,
- AUS per UI measurement,
- Configure panel,
- Filter/Limit Results panel,
- Gating panel,
- Hold Diff,
- Hold Diff measurement,
- measurement configuration,
- measurements tab,
- Reference Levels panel,
- Setup Diff,
- Setup Diff measurement,
- tCH(abs),
- tCH(abs) measurement,
- tCH(avg),
- tCH(avg) measurement,
- tCK(avg),
- tCK(avg) measurement,
- tCKSRE,
- tCKSRE measurement,
- tCKSRX measurement,
- tCL(abs),
- tCL(abs) measurement,
- tCL(avg),
- tCL(avg) measurement,
- tCMD-CMD,
- tCMD-CMDmeasurement,
- tDQSCK,
- tDQSCK measurement,
- tERR(m-n),
- tERR(m-n) measurement,
- tERR(n),
- tERR(n) measurement,
- timing measurements,
- tJIT(cc),
- tJIT(cc) measurement,
- tJIT(duty),
- tJIT(duty) measurement,
- tJIT(per),
- tJIT(per) measurement,
- tPST,
- tPSTmeasurement,
- tRPRE,
- tRPRE measurement,
- ttCKSRX,
- tWPRE,
- tWPRE measurement,
- Vix(ac),
- DDR measurement configuration menu, 269
- DDR Read & Write search,
- DDR Read search
  - DDR Input Configure menu,
  - DQ/DQS Levels Configure menu,
- DDR Read/Write search
  - DDR Input Configure menu,
  - DQ/DQS Levels Configure menu,
- DDR Write search
  - DDR Input Configure menu,
  - DQ/DQS Levels Configure menu,
- DDR, (Option 6-DBDDR3, SUP6-DBDDR3), 21
- decimal virtual logic keypad, 554
- declassification, 18
- Default setup, 132
- Default Setup button, 43
- define custom colors,
- define inputs (Setup and Hold trigger),
- Define Inputs menu (Logic search),
- define inputs, Logic trigger,
- define parallel bus inputs,
- definition,
- Delay,
- delay measurement,
- delay trigger, 579
- delete a measurement badge, 116
- delete files,
- Demo,
- deskew channels,

- deskew probes, 94, 95
- Deskew pulse generator accessory, 6
- diagnostics,
- digital acquisition, 137
- digital channel configuration, 392
- digital voltmeter, 393
- disable Autoset,
- disable Ethernet port (Opt 5-SEC),
- disable Ethernet port (Opt 6-SEC),
- disable firmware updates (Opt 5-SEC),
- disable firmware updates (Opt 6-SEC),
- disable LAN port (Opt 5-SEC),
- disable LAN port (Opt 6-SEC),
- disable USB ports (Opt 5-SEC),
- disable USB ports (Opt 6-SEC),
- disconnect a network drive, 89
- display
  - eye diagram, 120
- display a channel, 105
- display cursors, 130
- display cursors (spectrum view),
- display mode, 179
- display parameters, 179
- Display Port video output (rear panel), 56
- display settings,
- Displayed waveform, 570
- DJ, 623
- domain name,
- dragging waveform and measurement badges, 69
- draw visual trigger area on screen, 154
- draw-a-box button, 183
- Draw-a-Box button, 57
- Draw-a-Box button (Zoom), 74
- Dual Dirac Deterministic Jitter, 623
- dual-dirac random jitter,
- Duration N-Periods,
- DVI video output (rear panel), 56
- DVM, 393
- DVM option, 59
- dynamic range limit marker, 59
  
- E**
- e\* Scope HTTP Port,
- e\*Scope, 90
- Edge search menu,
- Edge trigger menu,
- edit an area (visual trigger)
  - change an area's, 158
  - change an area's height, 158
  - change an area's horizontal center, 158
  - change an area's logic true condition, 158
  - change an area's shape, 158
  - change an area's source, 158
  - change an area's vertical center, 158
  - change an area's width, 158
  - flip an area horizontally, 158
  - flip an area vertically, 158
  - rotate an area, 158
- edit menu, 406
- Edit Vertices panel (Visual Trigger), 555
- edit visual trigger area
  - add a vertex, 155
  - move a vertex, 155
  - remove a vertex, 155
  - resize, 155
  - rotate, 155
- eject USB devices,
- electrostatic damage, preventing, 97
- elements
  - math waveforms, 593
- energy plot, 125
- enter password (Opt 5-SEC),
- enter password (Opt 6-SEC),
- enter security password (Opt 5-SEC),
- enter security password (Opt 6-SEC),
- Envelope, 203
- envelope acquisition mode, 565
- Environment requirements, 31
- equation editor (math waveform),
- erase memory,
- erase memory (Opt 5-SEC),
- erase memory (Opt 6-SEC),
- erase nonvolatile memory,
- erase nonvolatile memory (Opt 5-SEC),
- erase nonvolatile memory (Opt 6-SEC),
- Errors,

- ESD, preventing, 97
- Ethernet bus trigger settings,
- Ethernet port (rear panel), 56
- Ethernet serial bus menu,
- Ethernet, connect to, 86
- excursion (spectrum view),
- expansion point, waveform, 59
- Explicit Clock,
- export log files,
- external
  - triggering, 154
- external monitor, 96
- eye diagram
  - display, 120
  - mask testing, 121
  - plot, 120
- eye diagram plot
  - load mask test file, 202
  - recall mask test file, 202
- eye diagram plot mask,
- eye diagram plot menu
  - Settings,
- Eye high, 626
- Eye low, 627
- eye mask test configuration menu, 397

## F

- F/2 measurement,
- F/4 measurement,
- F/8 measurement,
- factory calibration,
- factory settings, 132
- Fall slew rate,
- Fall Time,
- fall time trigger, 150
- Falling Slew Rate,
- Fast Acq button, 43
- Fast acquisitions, 132
- fast frame panel (acquisition badge),
- FastAcq, 394
- FastAcq acquisition mode, 565
- FastFrame acquisition mode, 565
- FastFrame concepts, 568
- FastFrames, 594

- FFT
  - process, 598
- FFT aliasing, 599
- FFT math waveforms, 122
- FFT windows,
- file menu, 406
- file utilities menu,
- filter
  - high pass, 191
  - low pass, 191
  - measurement, 191
- Filter/Limit Results panel,
- Filter/Limit Results panel (DDR),
- firmware, how to update, 81
- fixed graticule mode,
- Flattop2 window
  - defined, 601
- FlexChannel connectors (front panel), 43
- FlexChannel inputs, 54
- FlexRay bus trigger settings,
- FlexRay serial bus menu,
- flip an area (visual trigger), 158
- font color, 443
- Force button, 43
- Frequency,
- front cover, 29
- Front cover, 5
- front panel
  - Acquisition, 43
  - Autoset button, 43
  - Aux In, 43
  - Aux Trig, 43
  - Bus button (front panel), 43
  - Channel buttons (front panel), 43
  - Clear button, 43
  - Cursors button, 43
  - Default Setup, 43
  - description, 43
  - Fast Acq button, 43
  - FlexChannel connectors, 43
  - Force button, 43
  - High Res button, 43
  - Horizontal, 43
  - Level knob, 43

- Math button (front panel), 43
- Miscellaneous, 43
- Mode button, 43
- multipurpose knobs, 43
- Navigate buttons (front panel), 43
- Position knob, 43
- Position knob (horizontal), 43
- probe compensation connectors, 43
- Ref button (front panel), 43
- Run/Stop button, 43
- Scale knob, 43
- Scale knob (horizontal), 43
- Single/Seq button, 43
- Slope button, 43
- Touch Off button, 43
- Trigger, 43
- USB ports, 43
- Vertical, 43
- Zoom button (front panel), 43
- Zoom/Pan knobs (horizontal), 43
- front panel controls, 54
- functional test, 80

## G

- gating,
- Gating panel (DDR),
- Gaussian FFT window
  - definition, 602
- getting help, 3
- GPIB address,
- GPIB talk/listen address, 89
- graticule intensity, 129
- graticule intensity, setting, 559
- graticule style, 129, 181
- graticule style, setting, 559
- grouping badges and waveforms, 71

## H

- H Bar cursors, 390
- Hamming FFT window,
- Hamming window
  - defined, 604
- handle rotate, 30

- handles, analog and digital, 59
- Hanning FFT window,
- Hanning window
  - defined, 603
- Hard transit case, 6
- harmonics, 599
- Harmonics,
- Harmonics algorithms
  - harmonics algorithm,
- Harmonics Results table menu,
- Height@BER, 627
- help menu, 406
- Help system, 77
- hex virtual keypad, 554
- Hi Res
  - acquisition mode, 203
- high pass filter, 191
- High Res acquisition mode, 565
- High Res button, 43
- High Time,
- histogram plot, 119
- Histogram plot menu,
- Hold Diff measurement (DDR),
- Hold Time,
- holdoff, trigger, 153
- Horizontal acquisition
  - delay, 572
  - position, 572, 586
  - reference point, 586
  - window, 570
- Horizontal controls, 43
- horizontal measurement units,
- horizontal menu, open, 101
- horizontal mode, 394
- horizontal scale, 394
- horizontal settings, 394
- host name,
- how to
  - add a harmonics results table to the screen,
  - add a measurement, 112
  - add a measurement plot, 117
  - add a measurement results table to the screen,
  - add a note to the screen, 134
  - Add bus waveform, 111

- add label to a measurement,
- Add math waveform, 111
- Add reference waveform, 111
- add waveform to screen, 105
- change display settings, 129
- change graticule intensity, 129
- change graticule style, 129
- change measurement settings, 115
- change persistence, 129
- change waveform colors,
- change waveform intensity, 129
- change waveform interpolation, 129
- check power-on self tests results, 34
- compensate TPP0500B, TPP1000 probes, 83
- configure a results table,
- connect probes, 40
- connect to network, 86
- connect to PC using USB cable, 89
- Define Mask,
- delete a measurement, 116
- delete columns in a results table,
- delete individual measurements from a results table,
- display a spectrum trace, 166
- display cursors, 130
- download, install firmware, 81
- enable daylight saving time mode, 391
- load a reference waveform, 198
- load a Session file., 201
- move columns in a results table,
- open acquisition menu, 104
- open horizontal menu, 101
- quickly display waveform (Autoset), 100
- recall a reference waveform, 198
- recall a Session file., 201
- recall an instrument setup, 200
- remote access the oscilloscope (from Web), 90
- run signal path compensation (SPC), 82
- save a plot image to a file,
- save a report to a file, 196
- save a screen image, 193
- save a session, 197
- save a spectrum trace file, 194
- save a waveform file, 194
- save instrument settings, 195
- save plot data to a file,
- search on an event, 127
- set acquisition parameters, 104
- set clock format (12/24 hr), 79
- set custom waveform colors,
- set Date/Time badge on, off, 79
- set fall time trigger, 150
- set GPIB talk/listen address, 89
- set horizontal parameters, 101
- set logic trigger, 149
- set pulse width trigger, 146
- set rise time trigger, 150
- set setup/hold trigger, 150
- set time zone, 79, 391
- set timeout trigger, 147, 148
- set trigger holdoff, 153
- set trigger parameters, 102
- set window trigger, 148
- update firmware, 81
- use Autoset, 100
- use mouse with the UI, 75
- How to
  - change display mode (stacked, Overlay), 129
- I
- I vs.  $\int V$  plot, 123
- I vs.  $\int V$  plot menu,
- I/O,
- I2C bus trigger settings,
- I2C serial bus menu,
- I2S,
- I3C serial bus menu,
- idata, plot,
- image, plot,
- Impulse-response testing, 607
- inactive channel buttons, 59
- inductance, 123
- Inductance Curve plot menu,
- ink saver mode, 412
- Input

- resistance, 569
- termination, 569
- Input Capacitance algorithms
  - Input Capacitance algorithm,
- Input channel
  - trigger sources, 574
- input signal level requirements, 32
- Inrush Current algorithms
  - Inrush Current algorithm,
- install license,
- install option license
  - how to, 34, 40, 75, 79, 81–83, 86, 89, 90, 100–102, 104, 105, 111, 112, 115–117, 127, 129, 130, 134, 146–150, 153, 166, 193–198, 200, 201, 391
- installed options,
- installed probes,
- instrument settings
  - saving, 195
- instrument setup
  - recalling, 200
- intensity, graticule, 129
- intensity, waveform, 129
- interpolation, 180, 569
- Interpolation, 564
- Interpolation mode, setting, 559
- inverted screen color mode,
- IP address,

## J

- jitter analysis, advanced (option 5-DJA, SUP5-DJA), 20
- jitter analysis, advanced (option 6-DJA, SUP5-DJA), 20
- jitter measurements
  - Phase Noise,
  - TIE,
- jitter separation model,
- Jitter summary,
- JTF BW,

## K

- Kaiser-Bessel FFT window
  - defined, 605
- key features, xxi, xxii

- keyboard, 554
- keyboard, installing, 96
- keypad, 555
- knob A, 43
- knob B, 43

## L

- label, measurement,
- LAN,
- LAN port (rear panel), 56
- LAN reset,
- LAN Reset,
- LAN status,
- LAN status LEDs, 54
- LAN, connect to, 86
- Left Justified (LJ) audio bus,
- Level, 578
- Level knob, 43
- license file (option install),
- license file (option), 24
- license key uninstall,
- limits
  - measurement, 192
- LIN bus trigger settings,
- LIN serial bus menu,
- linked cursors, 390
- load
  - waveform, 198
- load a mask for an eye diagram plot,
- load an eye diagram mask file, 202
- load option license file,
- loading
  - files,
  - reference waveforms,
  - session files,
  - setup files,
- lock to bench or rack, 39
- log files, how to export,
- logic keypad, 554
- Logic search menu,
- Logic Search- Define Inputs menu,
- logic trigger, 149
- Logic trigger input settings,
- Loop BW,

low pas filter, 191  
low pass filter,  
Low time,  
Low Time,  
LXI, 90

## M

MAC address,  
Magnetic property  
    B-H curve, 123  
markers (spectrum view), 167  
markers (spectrum view), 173  
marking waveform events (search), 127  
mask badge configuration menu, 397  
mask hits badge, 121  
mask testing (eye diagram), 121  
Math button, 43  
math editor, 595  
Math equation, 591  
math syntax, 595  
Math waveform  
    differentiation, 596  
    elements, 593  
    guidelines, 594  
    interactions, 591  
    offset, 597  
    position, 597  
    scale, 597  
    sources, 593  
math waveform equation editor,  
math waveform menu, 397  
math waveforms  
    FFT, 122  
max hold trace (spectrum view),  
Maximum,  
Maximum signal level, 569  
Mean,  
Measure button, 57  
measurement  
    adding a label, 187  
    AOS (DDR),  
    AOS Per UI (DDR),  
    AUS (DDR),

    AUS Per tCK (DDR),  
    AUS per UI (DDR),  
    customizing, 187  
    Hold Diff (DDR),  
    plots, 117  
    reference levels, 188  
    Setup Diff (DDR),  
    tCH(abs) (DDR),  
    tCH(avg) (DDR),  
    tCK(avg) (DDR),  
    tCKSRE (DDR),  
    tCKSRX (DDR),  
    tCL(abs) (DDR),  
    tCL(avg) (DDR),  
    tCMD-CMD (DDR),  
    tDQSCK (DDR),  
    tERR(m-n) (DDR),  
    tERR(n) (DDR),  
    tJIT(cc) (DDR),  
    tJIT(duty) (DDR),  
    tJIT(per) (DDR),  
    tPST(DDR),  
    tRPRE (DDR),  
    tWPRE (DDR),  
    Vix(ac) (DDR),  
Measurement algorithm  
    bit amplitude,  
Measurement algorithms  
    amplitude,  
    AOS (DDR),  
    AOS Per tCK (DDR),  
    AOS Per UI (DDR),  
    AUS (DDR),  
    AUS Per tCK (DDR),  
    AUS Per UI (DDR),  
    base,  
    bit high,  
    bit low,  
    burst width,  
    cycle peak-to-peak,  
    DC common mode,  
    differential crossover,  
    Edge1, 589



---

Edge1Polarity, 590  
 Edge2, 589  
 Edge3, 589  
 end, 589  
 EndCycle, 590  
 frequency,  
 HighRef, 588  
 histogram method, 588  
 Hold Diff (DDR),  
 hysteresis, 589  
 integration algorithm,  
 LowRef, 588  
 maximum,  
 mean,  
 MidRef, 588  
 min-max method, 587  
 minimum,  
 missing samples, 590  
 N-Periods,  
 negative duty cycle,  
 negative overshoot,  
 negative pulse width,  
 out of range samples, 590  
 peak to peak,  
 period,  
 phase,  
 positive duty cycle,  
 positive overshoot,  
 positive width,  
 power quality algorithm,  
 record length, 589  
 rise time,  
 RMS,  
 Setup Diff (DDR),  
 start, 589  
 tCH(abs) (DDR),  
 tCH(avg) (DDR),  
 tCK(avg) (DDR),  
 tCKSRE (DDR),  
 tCKSRX (DDR),  
 tCL(abs) (DDR),  
 tCL(avg) (DDR),  
 tCMD-CMD (DDR),  
 tDQSCK (DDR),  
 tERR(m-n) (DDR),  
 tERR(n) (DDR),  
 tJIT(cc) (DDR),  
 tJIT(duty) (DDR),  
 tJIT(per) (DDR),  
 top, base, 587  
 TPOS, 590  
 tPST (DDR),  
 tRPRE (DDR),  
 TSOFF, 590  
 tWPRE (DDR),  
 variables, 587  
 Vix(ac) (DDR),  
 waveform record length, 590  
 measurement annotations,  
 measurement badge, 63  
 measurement badge, delete, 116  
 Measurement configuration menu, 228  
 measurement filters, 191  
 measurement gates  
     setting, 190  
 measurement interpolation mode,  
 measurement label,  
 measurement limits, 192  
 Measurement Name panel,  
 Measurement Results table,  
 Measurement Results table menu,  
 measurement units, horizontal,  
 measurements  
     AC RMS,  
     Amplitude,  
     Base,  
     Burst Width,  
     Data Rate,  
     Delay,  
     Duration N-Periods,  
     Fall Time,  
     Falling Slew Rate,  
     Frequency,  
     High Time,  
     Hold Time,  
     Low Time,

- Maximum,
- Mean,
- Minimum,
- Negative Duty Cycle,
- Negative Overshoot,
- Negative Pulse Width,
- Peak-To-Peak,
- Period,
- Phase,
- Phase Noise,
- Positive Duty Cycle,
- Positive Overshoot,
- Positive Pulse Width,
- Rise Time,
- Rising Slew Rate,
- RMS,
- Setup Time,
- Skew,
- TIE,
- Time Outside Level,
- Top,
- Unit Interval,

memory erase,

menu

- Add Measurements menu, 210
- DDR measurement configuration, 269
- math configuration,
- Reference waveform, 474

Menu bar, 57

menu panels, 72

menus, 72

MIL-STD-1553 bus trigger settings,

min hold trace (spectrum view),

Mini keyboard, 6

Minimum,

Miscellaneous controls, 43

Mode button (front panel), 43

monitor, connecting, 96

mount a network drive (standard instrument, 87

mount a network drive (Windows OS instrument), 88

mount network drive menu,

Mouse, 5

mouse touchscreen UI equivalents, 75

- mouse, installing, 96
- movable graticule mode,
- move a vertex (visual trigger), 155
- move cursors, 130
- moving badges, 69
- multipurpose knobs, 43

## N

- Navigate buttons (horizontal), 43
- navigation buttons, badges, 63
- Negative Duty Cycle,
- Negative Overshoot,
- Negative Pulse Width,
- network address,
- network drive (standard instrument
  - mount a network drive, 87
  - unmount a network drive, 89
- network drive (Windows 10 instrument
  - unmount a network drive, 89
- network drive (Windows OS instrument
  - mount a network drive, 88
- network, connect to, 86
- nonvolatile memory erase,
- Normal screen color mode,
- normal trace (spectrum view),
- note, 134
- Note button, 57
- notes, 443
- NPJ measurement, 624
- numeric keypad, 555
- Nyquist point, 599

## O

- octal virtual keypad, 554
- Offset
  - math offset and position, 597
- on screen keyboard, 554
- open acquisition menu, 104
- open horizontal menu, 101
- opening files,
- operating
  - altitude range, 31
  - humidity range, 31

- temperature range, 31
  - operating power requirements, 31
  - option details,
  - option license,
  - option license file, load,
  - option license, uninstall, 26
  - options
    - install an option, 24
    - option license file, 24
  - Options
    - 5-DJA, SUP5-DJA (advanced jitter analysis), 20
    - 5-DJA, SUP6-DJA (advanced jitter analysis), 20
    - 5-PWR, SUP5-PWR, 5-PS2 (adv. power analysis), 21
    - 5-SEC (enhanced instrument security), 18
    - 6-PWR, SUP6-PWR, 6-PS2 (adv. power analysis), 21
    - 6-SEC (enhanced instrument security), 18
    - advanced jitter analysis (option 5-DJA, SUP5-DJA), 20
    - advanced jitter analysis (option 6-DJA, SUP6-DJA), 20
    - Power (Option 5-PWR, SUP5-PWR, 5-PS2), 21
    - Power (Option 6-PWR, SUP6-PWR, 6-PS2), 21
  - Options, bandwidth, 11
  - Other drawer
    - Channel Settings menu,
  - overlay display mode, 179
  - overlay mode (waveforms), 129
  - Overlay mode, setting, 559
- ## P
- packing list, 5
  - palette (Fast Acq), 132
  - Pan, 74
  - panels, menu, 72
  - parallel bus
    - trigger, 151
  - parallel bus inputs,
  - parallel bus menu,
  - parallel bus search,
  - parallel bus trigger settings,
  - password, change (Opt 5-SEC),
  - password, change (Opt 6-SEC),
  - password, enter (Opt 5-SEC),
  - password, enter (Opt 6-SEC),
  - password, set (Opt 5-SEC),
  - password, set (Opt 6-SEC),
  - paste files,
  - Peak Detect
    - acquisition mode, 203
  - peak detect acquisition mode, 565
  - peak markers (spectrum view),
  - Peak-To-Peak,
  - Period,
  - persistence, 180
  - persistence, waveform, 129
  - Phase,
  - Phase noise, 624
  - pinching and handle rotate, 30
  - PJ, 625
  - PLL clock recovery,
  - PLL model,
  - plot
    - BH Curve, 123
    - Control Loop Response (Bode), 126
    - I vs.  $\int V$ , 123
    - Power Supply Rejection Ratio (PSRR), 126
    - PSRR, 126
    - safe operating area (SOA), 124
    - SOA, 124
    - spectrum, 119
    - time trend, 119
    - Trajectory, 124
    - XY, 120
    - XYZ, 120
  - Plot
    - bar graph, 125
    - energy, 125
    - histogram, 119
    - inductance, 123
    - power, 125
  - plot a measurement, 117
  - Plot button, 57
  - plot data file,
  - plot image file,
  - plot menu
    - Histogram,
  - plot menu, Trend,

- plot view configuration menus, 444
- Plot XY menu,
- plot XYZ menu,
- plots, adding, 306
- Position, 597
- Position control, 570
- Position knob, 43
- Position knob (horizontal), 43
- Positive Duty Cycle,
- Positive Overshoot,
- Positive Pulse Width,
- power, 21
- Power Autoset,
- Power cord, 5
- power cord connector (rear panel), 56
- power cord options, 22
- power cord wrap, 33
- Power measurement algorithms
  - frequency,
  - negative duty cycle,
  - negative pulse width,
  - period,
  - positive duty cycle,
  - positive width,
- Power Measurement Name panel,
- Power Measurements
  - Input analysis,
  - power autoset,
  - power preset,
- power plot, 125
- Power Preset,
- Power Quality,
- power requirements, 31
- power standby mode, 33
- Power Supply Rejection Ratio,
- Power Supply Rejection Ratio plot, 126
- Power Supply Rejection Ratio plot menu,
- power-on self tests,
- power-on test results, 34
- power, (Option 5-PWR, SUP5-PWR, 5-PS2), 21
- power, (Option 6-PWR, SUP6-PWR, 6-PS2), 21
- powering on or off, 33
- preventing ESD, 97

- probe
  - unsupported,
- probe comp connectors, 54
- probe compensation,
- probe compensation (TPP0500B, TPP1000), 83
- probe compensation connectors, 43
- Probe errors,
- probe inputs, 43
- probe setup
  - analog,
- probes, connecting, 40
- product description, xxi, xxii
- projector, connecting, 96
- proper handle rotate, 30
- Pulse Width search menu,
- pulse width trigger, 146
- Pulse width trigger,

## Q

- Q-factor, 628

## R

- rack mount kit information, 42
- Rackmount kit accessory, 6
- readout (spectrum view),
- rear panel
  - AFG Out, 56
  - AUX Out, 56
  - cable lock, 56
  - Display Port video output, 56
  - DVI video output, 56
  - Ethernet port (RJ-45), 56
  - LAN port (RJ-45), 56
  - power cord, 56
  - security cable lock, 56
  - USB Device port, 56
  - USB Host ports, 56
  - VGA video output, 56
  - video outputs, 56
- rear panel connections, 56
- recall
  - instrument setup, 200

- waveform, 198
  - recall an eye diagram mask file, 202
  - recall mask (SOA plot),
  - recall mask configuration file for eye diagram plot,
  - recall reference file menu,
  - recall spectrum trace file to spectrum view, 177
  - recalling, 193
  - record length, 394
  - Record length, 572
  - Record length memory options, 14
  - record view, waveform, 59
  - Rectangular FFT window,
  - Rectangular window
    - defined, 606
  - redo last action,
  - Ref button, 43
  - Ref In, 56
  - Ref In (rear panel), 56
  - reference clock out,
  - reference levels
    - measurement, 188
  - reference levels panel,
  - Reference Levels panel (DDR),
  - reference marker (spectrum view), 173
  - Reference waveform menu, 474
  - reference waveforms, 111
  - remote access (e\*Scope), 90
  - remote access (Web-based), 90
  - remove a vertex (visual trigger), 155
  - remove AC power from instrument, 33
  - rename files,
  - reports
    - saving, 196
  - requirements
    - altitude, 31
    - environment, 31
    - humidity, 31
    - power, 31
    - signal inputs, 32
    - temperature, 31
  - resize an area (visual trigger), 155
  - Results bar, 57
  - results table (bus),
  - Results Table button, 57
  - results tables,
  - return license,
  - right click settings,
  - Right Justified (RJ) audio bus,
  - Rise Time,
  - rise time trigger, 150
  - Rise/Fall Time search menu,
  - rise/fall time trigger,
  - Rising Slew Rate,
  - Rj, 625
  - RJ 66,
  - RJDIRAC,
  - RM5 rack mount, 42
  - RMS,
  - Rollmode acquisition mode, 565
  - rotate an area (visual trigger), 155, 158
  - RS232 bus trigger settings,
  - RS232 serial bus menu,
  - run signal path compensation, 82
  - Run/Stop
    - Acquisition menu, 203
  - Run/Stop button, 43
  - Runt search menu,
  - Runt trigger, 580
- ## S
- safe operating area plot, 124
  - Sample
    - acquisition mode, 203
  - sample acquisition mode, 565
  - sample interpolation, 569
  - sample rate, 394
  - Sampling process
    - defined, 563
  - sanitize, 18
  - save
    - instrument settings, 195
    - reports, 196
    - screen image, 193
    - sessions, 197
    - spectrum trace file, 194
    - waveform file, 194
  - save a plot image to a file,
  - Save As dialog,

- save bus decode results table to file,
- Save button, 43
- save files on trigger event,
- save harmonics results table to file
  - how to, 34, 40, 75, 79, 81–83, 86, 89, 90, 100–102, 104, 105, 111, 112, 115–117, 127, 129, 130, 134, 146–150, 153, 166, 193–198, 200, 201, 391
- save mask (SOA plot),
- save measurement results table to file
  - how to, 34, 40, 75, 79, 81–83, 86, 89, 90, 100–102, 104, 105, 111, 112, 115–117, 127, 129, 130, 134, 146–150, 153, 166, 193–198, 200, 201, 391
- Save menu action,
- Save on trigger
  - acquisition mode, 203
- save plot data to a file,
- save spectrum trace to a file, 177
- saving, 193
- saving ink on printed screen captures, 412
- scale buttons, badge, 63
- Scale controls
  - math, 597
  - positioning, 570
- Scale knob, 43
- Scale knob (horizontal), 43
- screen capture, 193
- screen captures, saving,
- screen color mode,
- screen dim,
- screen image
  - saving, 193
- screen keyboard, 554
- search,
- search badge, 63
- Search button, 57
- search menu
  - DDR Read,
  - DDR Read & Write,
  - DDR Write,
  - Edge,
  - Pulse Width,
  - Rise/Fall Time,
  - Runt,
  - Setup and Hold,
  - Timeout,
  - Window,
- Search menu
  - Logic,
- search serial bus,
- search tables and zoom mode, 185
- searching for events, 127
- security,
- security (Opt 5-SEC),
- security (Opt 6-SEC),
- security cable lock, 56
- self tests,
- SENT bus trigger settings,
- SENT serial bus menu,
- Sequence trigger menu,
- sequence trigger- A event,
- sequence trigger- B event,
- sequential acquisition mode, 203
- sequential triggering, 152
- Sequential triggering,
- serial bus
  - triggering, 151
- serial bus configuration, 272
- serial bus menu
  - SENT,
  - SPMI,
- serial bus search settings,
- serial bus trigger settings,
- serial bus, audio,
- serial bus, auto,
- serial bus, Ethernet,
- serial bus, FlexRay,
- serial bus, I2C,
- serial bus, I3C,
- serial bus, LIN,
- serial bus, RS232,
- serial bus, trigger options, 19
- service name,
- service options, 23
- Session file
  - how to load, 201
  - how to recall, 201
- sessions

- saving, 197
- set
  - clock format (12/24 hr), 79
  - Date/Time badge display on, off, 79
  - GPIO talk/listen address, 89
  - time zone, 79
- set custom waveform colors,
- set eye mask hits, 397
- set measurement gates, 190
- set password (Opt 5-SEC),
- set password (Opt 6-SEC),
- set security password (Opt 5-SEC),
- set security password (Opt 6-SEC),
- set trigger logic qualification inputs,
- setting up a bus, 272
- Settings bar, 57
- settings, Edge trigger,
- setup
  - parallel bus, 141
- Setup,
- Setup and Hold Search - Define Inputs menu,
- Setup and Hold search menu,
- Setup and hold trigger,
- Setup Diff measurement (DDR),
- Setup Time,
- setup/hold trigger, 150
- shared reference levels,
- shipped probes, 29
- show a measurement, 112
- signal input levels, 32
- signal path compensation,
- single acquisition mode, 203
- single sequence, 105
- Single Sequence/Stop After
  - Acquisition menu, 203
- Single/Seq
  - Acquisition menu, 203
- Single/Seq button, 43
- Skew,
- Slope
  - trigger, 578
- Slope button (front panel), 43
- SOA plot
  - recall mask,
  - save mask,
- SOA plot menu,
- Sources
  - math waveforms, 593
- SPC,
- SPC (signal path compensation), 82
- spectral FastAcq setting, 394
- spectrum plot, 119
- Spectrum Plot menu,
- Spectrum Time, 169
- spectrum view
  - automatic markers, 173
  - average trace mode,
  - channel badge, 175
  - concepts, 163
  - cursor display on-off,
  - cursor mode,
  - cursors configuration menu,
  - display mode,
  - excursion,
  - FFT versus Spectrum View, 163
  - front panel controls, 174
  - graticule intensity,
  - graticule style,
  - how to display a spectrum trace, 166
  - M, A, N, m traces, 167
  - markers, 167, 173
  - max hold trace mode,
  - menus, 478
  - min hold trace mode,
  - normal trace mode,
  - peak markers,
  - quantity (markers per waveform,
  - readout,
  - recall trace, 177
  - reference marker, 173
  - save trace to a file, 177
  - selected trace configuration menu,
  - source waveform,
  - spectrum badge, 175
  - spectrum trace file names, 177
  - spectrum view window configuration menu,
  - threshold,

- time-domain vs frequency domain, 163
- trace handle, 167
- trace mode,
- user interface, 167
- waveform intensity,
- Spectrum View
  - Spectrum Time Bar, 169
- spectrum view panel (channel menu),
- SPI bus trigger settings,
- SPI serial bus,
- split cursors, 390
- SPMI bus trigger settings,
- SPMI serial bus menu,
- SSC FREQ DEV,
- stacked display mode, 179
- stacked mode (waveforms), 129
- Stacked mode, setting, 559
- standard accessories, 5, 29
- statistic readouts on measurement badge, 228
- stop acquisitions, 203
- storage pouch, 29
- subnet mask,
- Support, 3
- Switching Loss
  - Trajectory, 124
- symbolic search config menu (CAN bus),
- syntax
  - math editor, 595
- system information,

## T

- tCH(abs) measurement (DDR),
- tCH(avg) measurement (DDR),
- tCK(avg) measurement (DDR),
- tCL(abs) measurement (DDR),
- tCL(avg) measurement (DDR),
- TDM audio bus,
- tDQSCK measurement (DDR),
- Technical support, 3
- Tek exponential FFT window
  - defined, 607
- TekExp FFT window, 607
- TekSecure,
- TekSecure memory erase,

- Tektronix Technical Support, 3
- TekVPI input connectors, 43
- TekVPI probes, 8
- Termination, 569
- tERR(m-n) measurement (DDR),
- tERR(n) measurement (DDR),
- text, 443
- threshold (spectrum view),
- TIE, 622
- time, 391
- Time outside level, 620
- Time Outside Level,
- time trend plot, 119
- time zone, 391
- time zone, how to set, 79
- Timebase Reference Source
  - Acquisition menu, 203
- Timeout search menu,
- timeout trigger, 147, 148
- Timeout trigger,
- timing measurements
  - Burst Width,
  - Data Rate,
  - Delay,
  - Duration N-Periods,
  - Fall Time,
  - Falling Slew Rate,
  - Frequency,
  - High Time,
  - Hold Time,
  - Low Time,
  - Negative Duty Cycle,
  - Negative Pulse Width,
  - Period,
  - Phase,
  - Positive Duty Cycle,
  - Positive Pulse Width,
  - Rise Time,
  - Rising Slew Rate,
  - Setup Time,
  - Skew,
  - Time Outside Level,
  - Unit Interval,



- TJ@BER, 625
  - tJIT(cc) measurement (DDR),
  - tJIT(duty) measurement (DDR),
  - tJIT(per) measurement (DDR),
  - Top,
  - Touch Off button, 43
  - touch screen tutorial,
  - touchscreen UI tasks, 75
  - TPP0500B, 5
  - TPP1000, 5
  - tPST measurement (DDR),
  - Trajectory plot, 124
  - Trajectory plot menu,
  - Transit case, hard plastic, 6
  - Trash Can icon, 57
  - Trend Plot menu,
  - trigger
    - external signal, 154
    - level indicators, 59
    - logic,
    - parallel bus, 151
    - position indicator, 59
    - pulse width,
    - rise/fall time,
    - serial bus, 151
    - timeout,
    - window,
  - Trigger
    - considerations, 572
    - logic qualification conditions,
    - modes, 576
    - slope and level, 578
    - sources, 574
  - Trigger controls, 43
  - trigger delay, 579
  - trigger event file save,
  - trigger holdoff, 153
  - trigger menu, 102
  - Trigger menus, 485
  - trigger out,
  - trigger the oscilloscope, 102
  - trigger types, 575
  - Triggering
    - advanced, 579
    - modes, 576
    - sources, 574
  - triggering concepts, 145, 573
  - tRPRE measurement (DDR),
  - turn instrument on or off, 33
  - tWPRE measurement (DDR),
- ## U
- undo last action,
  - uninstall an option license,
  - uninstall option license
    - how to, 34, 40, 75, 79, 81–83, 86, 89, 90,
    - 100–102, 104, 105, 111, 112, 115–117,
    - 127, 129, 130, 134, 146–150, 153, 166,
    - 193–198, 200, 201, 391
  - Unit Interval,
  - unmount a network drive, 89
  - unmount a network drive (Windows 10 instrument, 89
  - unmount USB devices,
  - unsupported probe,
  - USB,
  - USB bus trigger settings,
  - USB cable, connect to PC, 89
  - USB Device port (rear panel), 56
  - USB Host ports (rear panel), 56
  - USB ports (front panel), 43
  - USB serial bus menu,
  - use cursors, 130
  - user button,
  - user interface tutorial,
  - user preferences
    - define custom (waveform) colors,
    - define custom colors,
    - display settings,
    - graticule settings (movable or fixed),
    - using mouse with the touchscreen, 75
  - utility menu, 406
- ## V
- V Bar cursors, 390
  - V&H Bar cursors, 390
  - variables (math equation editor),

- Vertical acquisition, 571
- Vertical acquisition window, 570
- Vertical controls, 43
- Vertical offset, 571
- vertical settings, 379
- VGA video output (rear panel), 56
- video outputs (rear panel), 56
- virtual keyboard, 554
- virtual keypad, 554, 555
- visual trigger
  - change an area's height, 158
  - change an area's horizontal center, 158
  - change an area's logic true condition, 158
  - change an area's shape, 158
  - change an area's source, 158
  - change an area's vertical center, 158
  - change an area's width, 158
  - create a visual trigger area, 154
  - flip an area horizontally, 158
  - flip an area vertically, 158
  - rotate an area, 158
- Visual Trigger
  - Area configuration menu, 555
  - area settings panel, 555
  - Edit Vertices panel, 555
- visual trigger concepts,
- voltmeter, 393
- Volts per division
  - maximum, 569

## W

- Wave Inspector, 185
- waveform
  - expansion point, 59
  - intensity, 129
  - persistence, 129
  - recalling, 198
  - record view, 59
- waveform badge, 63
- waveform colors, how to change,
- waveform cursors, 390
- Waveform differentiation, 596
- waveform dots style, setting, 559
- waveform editor, 595

- waveform file
  - saving, 194
- Waveform integration, 597
- waveform intensity, setting, 559
- waveform interpolation, 180
- waveform memory options, 14
- waveform persistence, 180
- waveform persistence, setting, 559
- waveform preview, 586
- waveform style, setting, 559
- waveform vectors style, setting, 559
- Waveform View, 57
- Waveform View settings, 559
- Waveforms
  - math, 594
- waveforms, saving,
- Width@BER, 628
- Window search menu,
- window trigger, 148
- Windows 10 operating system (option 5-WIN, SUP5-WIN), 16
- Windows 10 operating system (option 6-WIN, SUP6-WIN), 16

## X

- XY plot, 120
- XY plot menu,
- XYZ plot, 120
- XYZ plot menu,

## Z

- zoom, 183, 185
- Zoom, 394
- Zoom box, 74
- Zoom button (front panel), 43
- zoom icon, 59
- zoom mode, 183
- Zoom overview, 74
- Zoom title bar, 74
- Zoom/Pan knobs (horizontal), 43
- zooming on search events, 185