Tektronix®

7 Series DPO Performance Oscilloscope

DPO714AX, DPO718AX

Help

Register now!

Click the following link to protect your product.

www.tek.com/register

077-1859-05 August 2026

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

TEKTRONIX and TEK are registered trademarks of Tektronix, Inc.

Supports Product Firmware

TekSecure is trademark of Tektronix, Inc.

These are the original instructions in English.

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



Contacting Tektronix

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

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

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

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

Contents

Welcome to the 7 Series Performance Oscilloscope embedded Help 20

Open Source GPL License Notice 20

Important safety information 21
- General safety summary 21
- To avoid fire or personal injury 21
- Probes and test leads 23
- Service safety summary 24
- Terms in this manual and on the product 24
- Symbols on the product 25
- Compliance information 25
- Safety compliance 25
- Environmental compliance 27
- Operating requirements and instrument connectors 28

Options 30
- Install option upgrade licenses 30
- Uninstall (return) an option license 31
- Drive names for system and USB memory devices 31

Getting acquainted with your instrument 33
- Installing and activating Windows 10 operating system 33
  - Install the SSD 33
  - Powering on Windows for the first time 33
  - Activating Windows 33
  - Differences between Windows and base instrument user interfaces 34
  - Updating the Windows TekScope application software 35
- Secure (lock) the instrument 35
- Connecting probes to the instrument 35
- Rackmount option information 36
- Front panel controls and connectors 36
- Rear panel connections 41
- User interface screen 41
- User interface elements 43
- Badges 46
  - Channel and Waveform badges 46
  - Measurement badges 47
  - SIM badges 47
  - Mask Test badge 48
  - Waveform Histogram badges 49
  - Cursor badges 49

Search badges . . . 50
Deleting mass Measurements/Searches badges at once . . . 50
Signal clipping and badges . . . 51
Error messages and badges . . . 51
System badges . . . 52
Common badge actions . . . 53
Badge selection status . . . 54
Moving waveform and measurement badges . . . 54
Group signal badges in the Settings bar . . . 56
Configuration menus . . . 56
Zoom user interface . . . 58
Using the touch screen interface for common tasks . . . 60
Accessing application help . . . 61
ESD Prevention Guidelines . . . 62
Cleaning, service, maintenance, and packing . . . 62

Configure the instrument . . . 63
Set the time zone and clock readout format . . . 63
Functional check . . . 63
Download and install the latest instrument firmware . . . 63
Run Signal Path Compensation (SPC) . . . 64
TriMode probe DC calibration . . . 65
Connect to a network (LAN) . . . 66
Mount a network drive from a standard instrument . . . 67
Mount a network drive on the Windows 10 instrument . . . 68
Mount TekDrive . . . 68
Unmount a network drive from a standard instrument . . . 69
Unmount a network drive from the Windows 10 instrument . . . 69
Unmount a TekDrive . . . 69
Connect the oscilloscope to a PC using a USB cable . . . 70
Remote access from a Web browser . . . 70
Remote access to a Windows 10 instrument . . . 71
Install TightVNC on the Windows 10 instrument . . . 72
Install TightVNC on a PC . . . 72
Deskew analog input channels . . . 73
Connect a keyboard or mouse . . . 76
Connect an external monitor or projector . . . 76
Calibrate the oscilloscope touchscreen when connected to external monitor . . . 77

Analog channel operating basics . . . 78
Acquiring a signal . . . 78
Add a callout to a view . . . 78
Delete a Callout . . . 80
Add a channel waveform to the display . . . 80
Add a measurement . . . 80
Add a math, reference, or bus waveform . . . 82
Add a plot of a measurement . . . 83

Add a Search . . . 86
Add mask testing to an Eye Diagram or BER Eye Contour plot . . . 88
Analog view of Bus . . . 89
Autoset to quickly display a waveform . . . 89
Change waveform view settings . . . 90
Configure a measurement . . . 91
Configure channel and waveform settings . . . 92
Delete a Measurement or Search badge . . . 97
Display a Acq Trend plot . . . 97
Display an Eye Diagram plot . . . 97
Display a Transfer Function plot . . . 98
Display an FFT math waveform . . . 98
Display a Histogram plot . . . 98
Display a Time Trend plot . . . 98
Display an XY or XYZ plot . . . 99
Display and configure cursors . . . 99
Display and configure Multi-source cursor readouts . . . 102
How to trigger on a signal . . . 103
Start and stop an acquisition . . . 105
Set the acquisition mode . . . 105
Set Horizontal parameters . . . 105
Using Default Setup . . . 106
Using FastAcq . . . 106

Setting waveform display parameters . . . 108
Set waveform display mode (Stacked or Overlay) . . . 108
Set the Waveform Interpolation mode . . . 108
Set the waveform persistence, style, and intensity . . . 108
Set the graticule style and intensity . . . 109

Zoom in on waveforms . . . 110
Turn on Zoom mode . . . 110
Using Wave Inspector front-panel controls for zoom . . . 112
Zoom mode and Searches . . . 112

Mask testing . . . 114
Create a Mask on waveforms . . . 114
Create a Mask on Eye diagram plot . . . 114
Edit mask segments on the screen . . . 115
Edit waveform mask segments using the Mask Definition menu . . . 117
Edit eye mask segments using the Mask Definition menu . . . 120

Signal Integrity Modeling (SIM) . . . 122
Key features . . . 122
Requirements and installation . . . 122
Model overview . . . 123
The Physical Model . . . 123

The Simulation Model 123
Simple De-embed workflow 124
Simple Embed workflow 124
Advanced workflow 125
Rx Equalization workflow 125
Create a SIM model 126
Configuring the blocks 127
Errors in blocks 131
Create a custom bandwidth limit filter 132
Plots 133
Examples of tasks and troubleshooting 135
De-embedding the cables 136
Embedding the cables 137
De-embedding and Embedding the cables using Advanced workflow 138
Stub embedding simulation 140
Stub de-embedding simulation 143

Waveform histogram 149
Create a waveform histogram 149
Edit histogram on the screen 149
Waveform Histogram guidelines 150

Waveform acquisition 151
Acquisition concepts 151
Acquisition modes 152
Fast Frame concepts 153
Waveform sample interpolation 154
Scaling and positioning 154
Vertical acquisition considerations 154
Horizontal acquisition considerations 155

Saving and recalling information 156
Save a screen image 156
Save a waveform to a file 156
Save instrument settings to a file 157
Save reports 158
Save sessions 159
Recall a reference waveform 159
Recall a Setup file 160
Recall a Session file 160
Open a mask test file for an Eye Diagram or BER Eye Contour plot 161

Menus and dialog boxes 163
Act On Event configuration menu 163
Acquisition configuration menu 164

Fast Frame Panel 165
Fast Frame Averaging 167
Fast Frame badge 168
Fast Frame badge configuration menu 169
Add Measurements configuration menu overview 170
Amplitude Measurements panel 171
Jitter Measurements panel 173
Jitter tab (Advanced Jitter and Eye Analysis) 173
Standard measurements tab 180
Time Measurements panel 180
Add Results Table configuration menu 184
Results Tables operations overview 184
Bus Decode Results table configuration menu 184
Annotation and navigation on waveform plots/data and results table 188
Frequency Response Analysis (FRA) Results table configuration menu 189
Navigation on Bar Graph and Harmonics Results Table 189
Measurement Results Table configuration menu 189
Peak Marker results table configuration menu 191
Search Results table menu 192
Save As configuration menu, Custom Results Table 194
AFG configuration menu 194
Analog Channel configuration menu 196
QuietChannel™ technology guidelines 198
Invert channel guidelines 200
Probe Setup panel (Channel configuration menu) 200
Unsupported Probe dialog 201
Probe Compensation configuration menu (analog channels Probe Setup panel) 202
Other panel (Channel configuration menu) 202
Deskew configuration menu (Other panel, Channel configuration menu) 203
Bus configuration menu 204
128b132b serial bus configuration menu 205
1-Wire serial bus configuration menu 206
64B/66B serial bus configuration menu 207
8b10b serial bus configuration menu 208
ARINC 429 serial bus menu 209
Audio serial bus configuration menu 211
Auto Ethernet serial bus configuration menu 212
CAN serial bus configuration menu 214
CPHY serial bus configuration menu 217
CXPI serial bus configuration menu 218
DPHY serial bus configuration menu 219
eSPI serial bus configuration menu 220
Ethernet serial bus menu 222
EtherCAT serial bus configuration menu 223
eUSB serial bus configuration menu 224
FlexRay serial bus configuration menu 226

I2C serial bus configuration menu 227
I3C serial bus configuration menu 228
LIN serial bus configuration menu 229
Manchester serial bus configuration menu 230
MDIO serial bus configuration menu 231
MIL-STD-1553 serial bus menu 232
NFC serial bus configuration menu 234
NRZ serial bus configuration menu 235
Parallel Bus configuration menu 237
Parallel Bus - Define Inputs menu 238
PCIe serial bus configuration menu 238
PSI5 serial bus configuration menu 240
RS232 serial bus menu 241
SDLC serial bus configuration menu 242
SENT serial bus configuration menu 243
SMBus serial bus configuration menu 244
SpaceWire serial bus configuration menu 245
SPI serial bus configuration menu 247
SPMI serial bus configuration menu 249
SVID serial bus configuration menu 250
USB serial bus configuration menu 251
Bus trigger radix-specific virtual keypads 253
Cursor configuration menu 254
Eye Mask Definition configuration menu 256
Date and Time configuration menu 259
Font Color menu (Text Settings configuration) 260
Horizontal configuration menu 260
Jitter measurement configuration menu overview 262
Jitter Measurement Name panel (Measurement configuration menu) 263
Configure panel (Jitter measurement configuration menu) 264
Reference Levels panel (Jitter measurement configuration menu) 285
Clock Recovery panel (Jitter measurement configuration menu) 285
Gating panel (Jitter measurement configuration menu) 286
Filter/Limit Results panel (Jitter measurement configuration menu) 286
Pass/Fail Testing panel (Jitter measurement configuration menu) 286
Mask Badge configuration menu 287
Right click menu functions associated with mask segments 287
Mask Definition configuration menu 287
Mask Test badge configuration menu 290
Right click menu functions associated with the Mask Test badge 292
Math configuration menu overview 293
Math configuration menu 293
Create/Edit filter configuration menu 295
Equation Editor (Math configuration menu) 297

Add Filter menu (math Equation Editor) . . . 298
Add Variable menu (math Equation Editor) . . . 298
Add Functions (math Equation Editor) . . . 299
Measurement configuration menu overview . . . 300
Measurement Name panel (Measurement configuration menu) . . . 300
Configure panel (Measurement configuration menu) . . . 301
Reference Levels panel (Measurement configuration menu) . . . 303
Clock Recovery panel (Measurement configuration menu) . . . 304
Clock Recovery- Advanced Settings configuration menu . . . 308
Gating panel (Measurement configuration menu) . . . 309
Filter/Limit Results panel (Measurement Settings menu) . . . 310
Pass/Fail Testing panel (Measurement Settings menu) . . . 311
Menu bar overview . . . 312
Recall configuration menu (File menu) . . . 313
Save As configuration menu (File menu) . . . 314
Browse Save As Location configuration menu . . . 318
File Utilities configuration (File menu) . . . 319
Mount Drive configuration menu . . . 321
Undo, Redo (Edit menu) . . . 322
Application (Menu bar) . . . 322
User Preferences (Utility menu) . . . 322
Define Custom Colors menu . . . 328
I/O (Utility menu) . . . 329
LAN and LAN SFP+ Reset configuration menu (Utility > I/O menu) . . . 334
Self Test configuration menu (Utility menu) . . . 335
Calibration configuration menu (Utility menu) . . . 336
Security configuration menu . . . 336
Enter Password configuration menu . . . 337
Set Password configuration menu . . . 338
Change Password configuration menu . . . 338
TekSecure Erase Memory . . . 339
Demo (Utility menu) . . . 339
Help... (Help menu) . . . 340
License Options...(Help menu) . . . 340
About (Help menu) . . . 340
Location to Save Exit Key configuration menu . . . 341
Browse License Files menu (Help > About) . . . 342
More (DRAW A BOX Menu) . . . 343
Add Plot configuration menu . . . 344
Acq (Acquisition) Trend plot configuration menu . . . 344
Bathtub plot configuration menu . . . 346
Composite Jitter Histogram (CJH) plot configuration menu . . . 348
Composite Noise Histogram (CNH) plot configuration menu . . . 349
Eye Diagram plot configuration menu . . . 352
Fast Frame Timestamp Trend plot configuration menu . . . 359
Histogram plot configuration menu . . . 360

Math FFT plot configuration menu (Math waveform) 362
Phase Noise plot configuration menu 364
Plot XY configuration menu 365
Save As configuration menu (plot Save panel, Save Plot Image button) 366
Save As configuration menu (plot Save panel, Save Plot Data button) 367
Time Trend Plot configuration menu 368
Transfer Function plot configuration menu 369
XYZ plot configuration menu 370
Reference waveform configuration menu 371
Recall configuration menu (Ref waveform configuration menu) 372
Remote channel configuration menu 373
Search configuration menu overview 374
Bus Search configuration menus 375
128b132b serial bus search configuration menu 375
1-Wire serial bus search configuration menu 376
64B/66B serial bus search configuration menu 377
8b10b serial bus search configuration menu 378
Audio serial bus search configuration menu 379
ARINC 429 serial bus search configuration menu 380
Auto Ethernet serial bus search configuration menu 382
CAN serial bus search configuration menu (when not using a .dbc symbol definition file) 386
CAN serial bus search configuration menu (when using a .dbc symbol definition file) 387
CAN DBC Symbolic Configuration menu 389
CPHY serial bus search configuration menu 390
CXPI serial bus search configuration menu 393
Configure custom pattern editor 394
Configure pattern editor 396
DDR Input Configure menu 397
DDR Read Search configuration menu 397
DDR Read & Write Search configuration menu 399
DDR Write Search configuration menu 401
DPHY serial bus search configuration menu 403
DQ/DQS Levels Configure menu 405
eSPI serial bus search configuration menu 405
Ethernet serial bus search configuration menu 412
EtherCAT serial bus search configuration menu 414
eUSB serial bus search configuration menu 418
FlexRay serial bus search configuration menu 422
I2C serial bus search configuration menu 424
I3C serial bus Search configuration menu 425
LIN serial bus search configuration menu 436
Manchester serial bus search configuration menu 437
MDIO serial bus search configuration menu 438
MIL-STD-1553 serial bus search configuration menu 441
NFC serial bus search configuration menu 442
NRZ serial bus search configuration menu 444

Parallel bus search configuration menu . . . 445
PCIe serial bus search configuration menu . . . 445
PSI5 serial bus search configuration menu . . . 449
RS-232 serial bus search configuration menu . . . 453
SDLC serial bus search configuration menu . . . 454
SENT serial bus search configuration menu . . . 457
SMBus serial bus search configuration menu . . . 459
SPI serial bus search configuration menu . . . 460
SPMI serial bus search configuration menu . . . 461
SpaceWire serial bus search configuration menu . . . 462
SVID serial bus search configuration menu . . . 464
USB serial bus search configuration menu . . . 465
Cycle Search configuration menu . . . 471
Edge Search configuration menu . . . 472
Logic search configuration menu . . . 473
Logic Search - Define Inputs configuration menu . . . 474
Pulse Width Search configuration menu . . . 475
Rise/Fall Time Search configuration menu . . . 476
Runt Search configuration menu . . . 477
Setup and Hold Search configuration menu . . . 478
Setup and Hold Search - Define Inputs configuration menu . . . 479
Timeout Search configuration menu . . . 480
Window Search configuration menu . . . 481
SIM configuration menu . . . 483
Block configuration menu . . . 486
Test Point configuration menu . . . 488
Test Point Save Filter configuration menu . . . 489
DUT SOURCE configuration menu . . . 489
File Viewer configuration menu . . . 491
Global Bandwidth Limit configuration menu . . . 491
Global Bandwidth Limit Save Filter configuration menu . . . 492
Advanced configuration menu . . . 492
Advanced Save configuration Menu . . . 493
Load configuration menu . . . 494
Probe configuration menu . . . 494
SAP configuration menu . . . 495
S-parameter Plot configuration menu . . . 496
Select Parameters configuration menu . . . 497
Scope configuration menu . . . 498
Receiver configuration menu . . . 498
Receiver Optimization Save Results configuration menu . . . 500
Receiver Parameters configuration menu . . . 500
Test point Math waveform configuration menu . . . 501
Clock Recovery configuration menu . . . 501
RX Filter configuration menu . . . 502

CTLE configuration menu 502
FFE/DFE configuration menu 502
FFE/DFE Save to File configuration menu 504
TDR measurement overview 504
Configure panel (TDR measurement configuration menu) 505
TDR Preset 505
Trigger configuration menu overview 507
Cycle Trigger configuration menu 507
Edge Trigger configuration menu 509
Pulse Width Trigger configuration menu 511
Logic Qualification - Define Inputs configuration menu 513
Rise/Fall Time Trigger configuration menu 513
Runt Trigger configuration menu 516
Sequence Trigger configuration menu 517
Timeout Trigger configuration menu 521
Error Detector configuration menu 522
Window Trigger configuration menu 526
Text Settings configuration menu (Callout and Waveform labels text) 528
Virtual Keyboard 529
Virtual Keypad 529
Visual Trigger panel 530
Visual Trigger Area configuration menu 530
Visual Trigger Area Combinatorial Logic menu 532
Waveform Histogram configuration menu 533
Waveform Histogram Result Badge menu 535
Save As configuration menu (Save Waveform Histogram) 535
Right click menu functions associated with the Waveform Histogram 536
Waveform View configuration menu 536

Triggering concepts 538
Trigger sources 539
Trigger types 539
Trigger modes 540
Trigger holdoff 540
Trigger slope and level 540
Trigger position in waveform record 540
Trigger delay 540
Cycle trigger concepts 541
Pulse width trigger concepts 541
Rise/Fall time trigger concepts 541
Runt trigger concepts 541
Sequential (A B) trigger concepts 541
Timeout trigger concepts 541
Window trigger 541
Visual Trigger concepts 542

Advanced triggering 544

Create Visual Trigger areas . . . 544
Edit visual trigger areas on the screen . . . 544
Edit visual trigger areas using the Area menu . . . 546
Set a Trigger Holdoff . . . 549
Trigger on a cycle event . . . 550
Trigger on a pulse width event . . . 552
Trigger on a rise or fall time event . . . 552
Trigger on a runt signal event . . . 554
Trigger on a sequential events (A and B triggers) . . . 555
Trigger on a signal window event . . . 556
Trigger on a timeout event . . . 556
Using the Error Detector . . . 557
Trigger on an external signal using the AUX input . . . 559

Measurement configuration . . . 560
Label a measurement . . . 560
Set measurement reference levels . . . 561
Set measurement gates . . . 562
Set measurement filters . . . 562
Set measurement limits . . . 563

Measurement concepts . . . 565
Measurement variables . . . 565
Definition of Base and Top . . . 565
Base, Top calculation methods . . . 565
HighRef, MidRef, LowRef . . . 566
Other variables . . . 566
Edge calculations . . . 566
Missing or out-of-range samples . . . 567
Math waveforms . . . 567
Math waveform elements . . . 568
Math waveform sources . . . 568
Guidelines for working with math waveforms . . . 568
Math waveform editor syntax . . . 569
Math waveform differentiation . . . 569
Math waveform offset, position, and scale . . . 570
Waveform integration . . . 570
FFT process . . . 571
FFT and aliasing . . . 571
Blackman-Harris FFT window concepts . . . 572
Flattop2 window . . . 572
Gaussian window . . . 573
Hanning FFT window . . . 573
Hamming window . . . 573
Kaiser-Bessel FFT window . . . 573

Rectangular window . . . 574
Tek-Exponential window . . . 574
User Defined Filter . . . 574

Measurement algorithms . . . 587
Amplitude measurement algorithms . . . 587
AC RMS measurement algorithm . . . 587
Area measurement algorithm . . . 587
Amplitude measurement algorithm . . . 587
Base measurement algorithm . . . 587
Integration algorithm . . . 587
Maximum measurement algorithm . . . 588
Mean measurement algorithm . . . 588
Minimum measurement algorithm . . . 588
Negative Overshoot measurement algorithm . . . 588
Positive Overshoot measurement algorithm . . . 588
Peak-To-Peak measurement algorithm . . . 589
RMS measurement algorithm . . . 589
Top measurement algorithm . . . 589
Eye measurement algorithms . . . 589
Eye Height measurement algorithm . . . 589
Eye high . . . 589
Eye low . . . 590
Eye Width measurement algorithm . . . 590
Height@BER . . . 590
Q-factor . . . 590
Width@BER . . . 591
Jitter measurement algorithms . . . 591
AC Common Mode . . . 591
Bit Amplitude measurement algorithm . . . 591
Bit High measurement algorithm . . . 591
Bit Low measurement algorithm . . . 591
DC Common Mode measurement algorithm . . . 591
Differential Crossover measurement algorithm . . . 591
SSC Freq Dev measurement algorithm . . . 591
SSC Modulation Rate measurement algorithm . . . 592
TIE . . . 592
T/nT Ratio measurement algorithm . . . 592
DCD . . . 592
DDJ . . . 592
DJ . . . 593
Dual Dirac deterministic jitter . . . 593
F/2 measurement algorithm . . . 593
F/4 measurement algorithm . . . 593
F/8 measurement algorithm . . . 593
J2 . . . 593

J9 . . . 593
CC-Jitter (Cycle to Cycle Jitter) . . . 593
Jitter Summary measurement . . . 593
NPJ . . . 594
Phase noise . . . 594
PJ . . . 594
SJ@Freq measurement algorithm . . . 594
RJ . . . 594
RJ δδ measurement algorithm . . . 594
SRJ . . . 594
TJ@BER . . . 595
Transfer Function algorithm . . . 595
Jitter Analysis Method . . . 595
Breakdown of jitter (Jitter map) . . . 595
Breakdown of noise (Noise Map) . . . 596
Jitter separation . . . 598
Joint Jitter/Noise analysis . . . 600
Timing measurement algorithms . . . 602
Burst Width measurement algorithm . . . 602
Data Rate measurement algorithm . . . 602
Delay measurement algorithm . . . 602
Falling slew rate . . . 602
Fall Time measurement algorithm . . . 603
Frequency measurement algorithm . . . 603
High Time measurement algorithm . . . 604
Hold Time measurement algorithm . . . 604
Low Time measurement algorithm . . . 604
N-Periods Duration measurement algorithm . . . 604
Negative Duty Cycle measurement algorithm . . . 605
Negative Pulse Width measurement algorithm . . . 605
Period measurement algorithm . . . 605
Phase measurement algorithm . . . 605
Positive Duty Cycle measurement algorithm . . . 606
Positive Pulse Width measurement algorithm . . . 606
Rising Slew Rate measurement algorithm . . . 606
Rise Time measurement algorithm . . . 606
Setup . . . 607
Skew . . . 607
Time Outside Level measurement algorithm . . . 608

Time to Max . . . 608
Time to Min . . . 608
Unit Interval measurement algorithm . . . 608
TDR measurement algorithm . . . 608
TDR preset algorithm . . . 610

Waveform display concepts . . . 610
Waveform preview mode . . . 611
Horizontal position and the horizontal reference point . . . 611

References . . . 612
SIM badge error and warning messages . . . 612
TDR badge error and warning messages . . . 613
User defined filter errors and warnings . . . 615

List of Figures

Connecting probes to the 7 Series DPO . . . 36
7 Series DPO controls . . . 37
Physical Model example . . . 123
Simulation Model example . . . 124
Stub embedding simulation model . . . 142
Stub de-embedding physical model . . . 144
Stub de-embedding simulation model . . . 147
Connection diagram for TDR single-ended . . . 506
Connection diagram for TDR differential . . . 506
Hanning window . . . 573
Hanning window side lobe . . . 573
Jitter map - Spectral Only . . . 596
Jitter map - Spectral + BUJ . . . 596
Noise model component interrelationships . . . 602
Fall Time . . . 603
Rho calibration regions . . . 610

List of Tables

Welcome to the 7 Series Performance Oscilloscope embedded Help

This Help supports the 7 Series instruments. See the following lists for information on the key features of each instrument. Use the tabs on the left to go to specific topics or search for a topic of interest.

Key features and benefits

- Bandwidths from 8 GHz - 25 GHz
- DPO714AX: 4 channels with TekConnect inputs
- DPO718AX: 4 channels with TekConnect input channels and 4 channels with 2.92 mm inputs channels
- Compatible with TekConnect probes and adapters
- 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 125 GS/s sample rate
- 500 M points record length on all channels (optional 1 G, and 2 G record lengths available)
- QuietChannel™ technology
- Cursors analysis: Waveform, V Bars, H Bars, V&H Bars, Multi source cursor (readouts)
- No set limit on the number of math, reference, and bus waveforms you can display (the number of waveforms depends on available system memory)
- Advanced serial bus triggering and analysis options let you decode and search on industry standard buses
- Integrated optional features include a 100 MHz arbitrary function generator (AFG) and trigger frequency counter
- Available measurement options provide additional measurement and analysis functions.
- Optional Signal Integrity Model (SIM) feature offers the capability to emulate the serial data channel and de-embed fixtures, cables, or probes.

Open Source GPL License Notice

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

Chief Intellectual Property Counsel, Tektronix, Inc.

MS 50/LAW

13725 SW Karl Braun Dr.

Beaverton OR, 97077

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

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

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

Important safety information

This manual contains information and warnings that must be followed by the user for safe operation and to keep the product in a safe condition.

To safely perform service on this product, see the *Service safety summary* that follows the *General safety summary*.

General safety summary

Use the product only as specified. Review the following safety precautions to avoid injury and prevent damage to this product or any products connected to it. Carefully read all instructions. Retain these instructions for future reference.

This product shall be used in accordance with local and national codes.

For correct and safe operation of the product, it is essential that you follow generally accepted safety procedures in addition to the safety precautions specified in this manual.

The product is designed to be used by trained personnel only.

Only qualified personnel who are aware of the hazards involved should remove the cover for repair, maintenance, or adjustment.

Before use, always check the product with a known source to be sure it is operating correctly.

This product is not intended for detection of hazardous voltages.

Use personal protective equipment to prevent shock and arc blast injury where hazardous live conductors are exposed.

While using this product, you may need to access other parts of a larger system. Read the safety sections of the other component manuals for warnings and cautions related to operating the system.

When incorporating this equipment into a system, the safety of that system is the responsibility of the assembler of the system.

To avoid fire or personal injury

Use proper power cord

Use only the power cord specified for this product and certified for the country of use.

Ground the product

This product is grounded through the grounding conductor of the power cord. To avoid electric shock, the grounding conductor must be connected to earth ground. Before making connections to the input or output terminals of the product, ensure that the product is properly grounded. Do not disable the power cord grounding connection.

Power disconnect

The power cord disconnects the product from the power source. See instructions for the location. Do not position the equipment so that it is difficult to operate the power cord; it must remain accessible to the user at all times to allow for quick disconnection if needed.

Use proper AC adapter

Use only the AC adapter specified for this product.

Connect and disconnect properly

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

Use only insulated voltage probes, test leads, and adapters supplied with the product, or indicated by Tektronix to be suitable for the product.

Connect the probe output to the measurement instrument before connecting the probe to the circuit under test. Connect the probe reference lead to the circuit under test before connecting the probe input. Disconnect the probe input and the probe reference lead from the circuit under test before disconnecting the probe from the measurement instrument.

De-energize the circuit under test before connecting or disconnecting the probe.

Observe all terminal ratings

To avoid fire or shock hazard, observe all rating and markings on the product. Consult the product manual for further ratings information before making connections to the product.

Do not exceed the Measurement Category (CAT) rating and voltage or current rating of the lowest rated individual component of a product, probe, or accessory. Use caution when using 1:1 test leads because the probe tip voltage is directly transmitted to the product.

Do not apply a potential to any terminal, including the common terminal, that exceeds the maximum rating of that terminal.

Do not float the common terminal above the rated voltage for that terminal.

The measuring terminals on this product are not rated for connection to mains or Category II, III, or IV circuits.

The measurement terminals on this product are not rated for connection to Category III or IV circuits.

Do not connect a current probe to any wire that carries voltages above the current probe voltage rating.

Do not operate without covers

Do not operate this product with covers or panels removed, or with the case open. Hazardous voltage exposure is possible.

Avoid exposed circuitry

Do not touch exposed connections and components when power is present.

Do not operate with suspected failures

If you suspect that there is damage to this product, have it inspected by qualified service personnel.

Disable the product if it is damaged. Do not use the product if it is damaged or operates incorrectly. If in doubt about safety of the product, turn it off and disconnect the power cord. Clearly mark the product to prevent its further operation.

Before use, inspect voltage probes, test leads, and accessories for mechanical damage and replace when damaged. Do not use probes or test leads if they are damaged, if there is exposed metal, or if a wear indicator shows.

Examine the exterior of the product before you use it. Look for cracks or missing pieces.

Use only specified replacement parts.

Wear eye protection

Wear eye protection if exposure to high-intensity rays or laser radiation exists.

Do not operate in wet/damp conditions

Be aware that condensation may occur if a unit is moved from a cold to a warm environment.

Do not operate in an explosive atmosphere

Keep product surfaces clean and dry

Remove the input signals before you clean the product.

Provide proper ventilation

Refer to the installation instructions in the manual for details on installing the product so it has proper ventilation.

Slots and openings are provided for ventilation and should never be covered or otherwise obstructed. Do not push objects into any of the openings.

Provide a safe working environment

Always place the product in a location convenient for viewing the display and indicators.

Avoid improper or prolonged use of keyboards, pointers, and button pads. Improper or prolonged keyboard or pointer use may result in serious injury.

Be sure your work area meets applicable ergonomic standards. Consult with an ergonomics professional to avoid stress injuries.

Use care when lifting and carrying the product. This product is provided with a handle or handles for lifting and carrying.

WARNING: The product is heavy. To reduce the risk of personal injury or damage to the device get help when lifting or carrying the product.

Use only the Tektronix rackmount hardware specified for this product.

Probes and test leads

Before connecting probes or test leads, connect the power cord from the power connector to a properly grounded power outlet.

Keep fingers behind the protective barrier, protective finger guard, or tactile indicator on the probes. Remove all probes, test leads and accessories that are not in use.

Use only correct Measurement Category (CAT), voltage, temperature, altitude, and amperage rated probes, test leads, and adapters for any measurement.

Beware of high voltages

Understand the voltage ratings for the probe you are using and do not exceed those ratings. Two ratings are important to know and understand:

- The maximum measurement voltage from the probe tip to the probe reference lead.
- The maximum floating voltage from the probe reference lead to earth ground.

These two voltage ratings depend on the probe and your application. Refer to the Specifications section of the manual for more information.

WARNING: To prevent electrical shock, do not exceed the maximum measurement or maximum floating voltage for the oscilloscope input BNC connector, probe tip, or probe reference lead.

Connect and disconnect properly.

Connect the probe output to the measurement product before connecting the probe to the circuit under test. Connect the probe reference lead to the circuit under test before connecting the probe input. Disconnect the probe input and the probe reference lead from the circuit under test before disconnecting the probe from the measurement product.

De-energize the circuit under test before connecting or disconnecting the current probe.

Connect the probe reference lead to earth ground only.

Do not connect a current probe to any wire that carries voltages or frequencies above the current probe voltage rating.

Inspect the probe and accessories

Before each use, inspect probe and accessories for damage (cuts, tears, or defects in the probe body, accessories, or cable jacket). Do not use if damaged.

Ground-referenced oscilloscope use

Do not float the reference lead of the probe when using with ground-referenced oscilloscopes. The reference lead must be connected to earth potential (0 V).

Floating measurement use

Do not float the reference lead of the probe above the rated float voltage.

Service safety summary

The *Service safety summary* section contains additional information required to safely perform service on the product. Only qualified personnel should perform service procedures. Read this *Service safety summary* and the *General safety summary* before performing any service procedures.

To avoid electric shock

Do not touch exposed connections.

Do not service alone

Do not perform internal service or adjustments of this product unless another person capable of rendering first aid and resuscitation is present.

Disconnect power

To avoid electric shock, switch off the product power and disconnect the power cord from the mains power before removing any covers or panels, or opening the case for servicing.

Use care when servicing with power on

Dangerous voltages or currents may exist in this product. Disconnect power, remove battery (if applicable), and disconnect test leads before removing protective panels, soldering, or replacing components.

Verify safety after repair

Always recheck ground continuity and mains dielectric strength after performing a repair.

Terms in this manual and on the product

These terms may appear in this manual:

WARNING: Warning statements identify conditions or practices that could result in injury or loss of life.

CAUTION: Caution statements identify conditions or practices that could result in damage to this product or other property.

These terms may appear on the product:

- DANGER indicates an injury hazard immediately accessible as you read the marking.
- WARNING indicates an injury hazard not immediately accessible as you read the marking.
- CAUTION indicates a hazard to property including the product.

Symbols on the product



When this symbol is marked on the product, be sure to consult the manual to find out the nature of the potential hazards and any actions which have to be taken to avoid them. (This symbol may also be used to refer the user to ratings in the manual.)

The following symbols may appear on the product.

 CAUTION: Refer to Manual	 Protective Ground (Earth) Terminal	 Earth Terminal	 Chassis Ground	 Do not connect to or remove from an uninsulated conductor that is HAZARDOUS LIVE.	
 WARNING: High Voltage	 Standby	 Functional Earth Terminal	 Use only on an insulated wire.		

Compliance information

This section lists the safety and environmental standards with which the instrument complies. This product is intended for use by professionals and trained personnel only; it is not designed for use in households or by children.

Compliance questions may be directed to the following address: Tektronix, Inc. PO Box 500, MS 19-045 Beaverton, OR 97077, US tek.com

Safety compliance

This section lists the safety standards with which the product complies and other safety compliance information.

EU declaration of conformity – low voltage

Compliance was demonstrated to the following specification as listed in the Official Journal of the European Union:

Low Voltage Directive 2014/35/EU.

- EN 61010-1. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 1: General Requirements
- EN 61010-2-030. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 2-030: Particular requirements for testing and measuring circuits

U.S. nationally recognized testing laboratory listing

- UL 61010-1. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 1: General Requirements
- UL 61010-2-030. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 2-030: Particular requirements for testing and measuring circuits

Canadian certification

- CAN/CSA-C22.2 No. 61010-1. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 1: General Requirements
- CAN/CSA-C22.2 No. 61010-2-030. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 2-030: Particular requirements for testing and measuring circuits

Additional compliances

- IEC 61010-1. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 1: General Requirements
- IEC 61010-2-030. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 2-030: Particular requirements for testing and measuring circuits

Equipment type

Test and measuring equipment.

Safety class

Class 1 – grounded product.

Safety certification of plug-in or VXI modules

The safety certification is valid only when installed in an appropriately approved (by a USA NRTL or a Canada Certified Organization) mainframe.

Pollution degree description

A measure of the contaminants that could occur in the environment around and within a product. Typically the internal environment inside a product is considered to be the same as the external. Products should be used only in the environment for which they are rated.

- Pollution Degree 1. No pollution or only dry, nonconductive pollution occurs. Products in this category are generally encapsulated, hermetically sealed, or located in clean rooms.
- Pollution Degree 2. Normally only dry, nonconductive pollution occurs. Occasionally a temporary conductivity that is caused by condensation must be expected. This location is a typical office/home environment. Temporary condensation occurs only when the product is out of service.
- Pollution Degree 3. Conductive pollution, or dry, nonconductive pollution that becomes conductive due to condensation. These are sheltered locations where neither temperature nor humidity is controlled. The area is protected from direct sunshine, rain, or direct wind.
- Pollution Degree 4. Pollution that generates persistent conductivity through conductive dust, rain, or snow. Typical outdoor locations.

Pollution degree rating

2

Measurement and overvoltage category descriptions

Measurement terminals on this product may be rated for measuring mains voltages from one or more of the following categories (see specific ratings marked on the product and in the manual).

- Overvoltage Category I. For equipment intended to be connected to a mains supply in which means have been taken to substantially and reliably reduce transient overvoltages to a level where they cannot cause a hazard.
- Measurement Category II. For measurements performed on circuits directly connected to the low-voltage installation.
- Measurement Category III. For measurements performed in the building installation.
- Measurement Category IV. For measurements performed at the source of low-voltage installation.

NOTE: Only mains power supply circuits have an overvoltage category rating. Only measurement circuits have a measurement category rating. Other circuits within the product do not have either rating.

Mains overvoltage category rating

II

Environmental compliance

This section provides information about the environmental impact of the product.

Product end-of-life handling

Observe the following guidelines when recycling an instrument or component:

Equipment recycling

Production of this equipment required the extraction and use of natural resources. The equipment may contain substances that could be harmful to the environment or human health if improperly handled at the product's end of life. To avoid release of such substances into the environment and to reduce the use of natural resources, we encourage you to recycle this product in an appropriate system that will ensure that most of the materials are reused or recycled appropriately.



This symbol indicates that this product complies with the applicable European Union requirements according to Directives 2012/19/EU and 2006/66/EC on waste electrical and electronic equipment (WEEE) and batteries. For information about recycling options, check the Tektronix Web site (https://www.tek.com/productrecycle).

Battery recycling

This product contains a small installed lithium metal button cell. Please properly dispose of or recycle the cell at its end of life according to local government regulations.

Perchlorate materials

This product contains one or more type CR lithium batteries. According to the state of California, CR lithium batteries are classified as perchlorate materials and require special handling. See www.dtsc.ca.gov/hazardouswaste/perchlorate for additional information.

Transporting batteries

The small lithium primary button cell contained in this equipment does not exceed 1 gram of lithium metal content per cell.

The cell type has been shown by the manufacturer to comply with the applicable requirements of the UN Manual of Tests and Criteria Part III, Sub-section 38.3. Consult your carrier to determine which lithium battery transportation requirements are applicable to your configuration, including to its re-packaging and re-labeling, prior to reshipment of the product by any mode of transport.

Operating requirements and instrument connectors

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

Power requirements

Power source voltage: 100 V - 240 V

Power source frequency: 50/60 Hz

Power consumption: 1600 W maximum

Environmental requirements

Temperature:

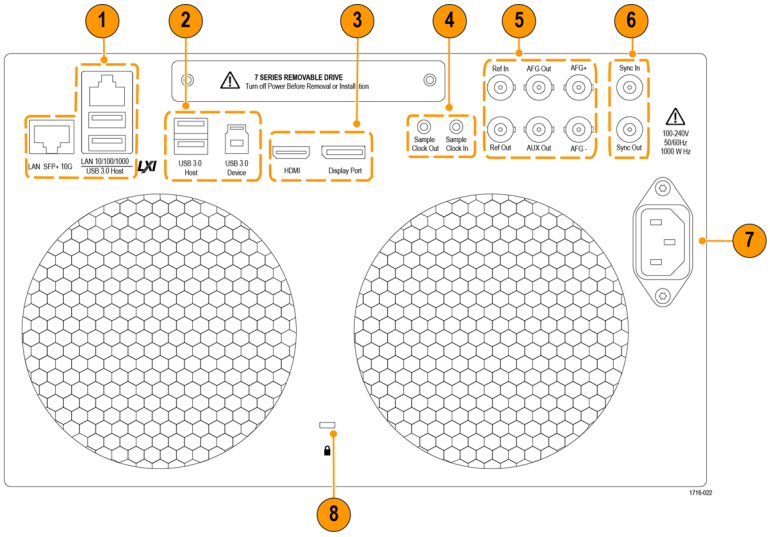
- **Operating**: 5 °C to +40 °C (+41 °F to +104 °F). For proper cooling, maintain the following clearances for the instrument: sides (60 mm (2.4 in), rear fan exhaust area (27 mm (1.1 in), front bottom (20 mm (0.8 in)).
- **Non-operating**: -20 °C to +60 °C (-4 °F to +140 °F).

Humidity:

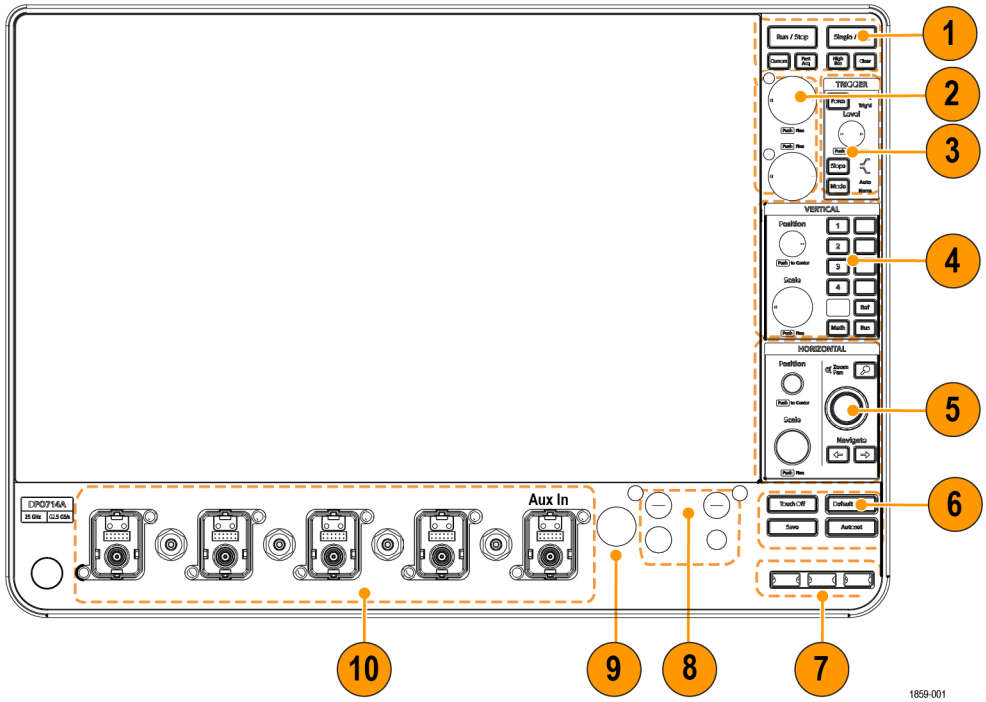
- **Operating**: 5% to 90% relative humidity at temperatures up to +40 °C.
- **Non-operating**: 5% to 90% relative humidity at temperatures up to +60 °C, non-condensing, and as limited by a maximum wet-bulb temperature of +39 °C.

Altitude:

- **Operating**: Up to 3000 meters (9842 feet)
- **Non-operating**: Up to 12000 meters (39370 feet)



1. LAN SFP+, LAN	**5.** Sample Clock Out, Sample Clock In
2. USB 3.0 Host, USB 3.0 Device	**6.** Ref In, Ref Out; Synch In, Synch Out; AFG Out, AFG +, AFG -; AUX Out
3. HDMI, Display Port	**7.** Power cord
4. For factory use only (MFG port)	



1. Acquisition and Cursor controls	**6.** Miscellaneous controls
2. Multi-purpose knobs	**7.** USB 3.0 host ports
3. Trigger controls	**8.** Differential Fast Edge for deskew; Ground antistatic connection
4. Vertical controls	**9.** DC probe calibration
5. Horizontal controls	**10. DPO714AX/DPO718AX:** TekConnect input channel probe connectors (4) and TekConnect AUX In connector (1); **DPO718AX only:** 2.92 mm Planar Crown® input channels (4)

Options

Some features described in this Help system are not supported by all product families, models, or configurations. Please refer to the datasheet for your specific product at tek.com to see which features it supports.

Install option upgrade licenses

Option license upgrades are field-installable licenses that you can purchase after receiving your instrument to add features to your oscilloscope. You install option upgrades by installing license files on the oscilloscope. Each option requires a separate license file.

Before you begin

These instructions do not pertain to options that were purchased and pre-installed on your instrument when ordered.

A node-locked license is valid only for the specific model number and serial number of the instrument for which it was purchased; it will not work on any other instrument. The single license file does not affect options that were factory installed or any other upgrades that you may have already purchased and installed.

Perpetual licenses allow you to keep the latest version you had installed even if license timeframe ends. Node-locked can be installed in factory or by user. Floating is only installed by user. Permanent licenses are installed in factory.

Floating licenses offer an alternative method to manage your Tektronix asset. Floating licenses allow license-key enabled options to be easily moved among all your 7 Series oscilloscopes. Access the Tektronix AMS at www.tek.com/products/product-license.

About this task

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.

Procedure

1. Follow the instructions you received to download the upgrade license file (<filename>.lic).
2. Copy the license file or files to a USB memory device.
3. Insert the USB drive into the powered-on oscilloscope for which the upgrade was purchased.
4. Select **Help** > **About**.
5. Select **Install License** to open the Browse License Files dialog box.
6. Find and select the upgrade license file to install.
7. Select **Open**. The oscilloscope installs the license and returns to the About screen. Verify that the installed license was added to the Installed Options list.
8. Repeat steps 5 through 7 for each upgrade license file that you purchased and downloaded.
9. Power cycle the oscilloscope to enable the installed upgrades.
10. If you installed a bandwidth upgrade, rerun signal path compensation (SPC). Then carefully remove the model/bandwidth label from the lower-left corner of the front panel and install the new model/bandwidth label that was sent through normal mail channels as part of the upgrade purchase.

Uninstall (return) an option license

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

Before you begin

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.

About this task

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.

Procedure

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.
   -
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.

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
DPO714AX, DPO718AX		

Drive name	Drive letter	Drive or physical USB port location
Root drive	**C**	User-accessible memory on the oscilloscope.
Front panel	**E**	USB 3.0
	F	
	G	
Rear panel	**H**	USB 3.0 (Host)
	I	USB 3.0 (Host)
	J	USB 3.0 (Host)
	K	USB 3.0 (Host)
Instruments with Windows OS		
Root drive	**C**	User-accessible memory on the oscilloscope.
USB ports	**Dynamic port letter assignment**	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.

Getting acquainted with your instrument

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

Installing and activating Windows 10 operating system

Use the following instructions to install the embedded operating system solid state drive or to install and activate Windows 10 on the instrument.

Install the SSD

All user-storable data is stored on the removable drive. Sanitize or replace the drive in a manner that meets local and any other security requirements that you have.

NOTE: To see the size of the drive, refer to the (C:) local disk properties window.

NOTE: Be sure that you create OS Restore media before scrubbing the hard drive/solid state drive or moving the instrument into a secure area.

After the drive is sanitized or replaced, the operating system and instrument software can be reinstalled. Following reinstallation of the operating system and product software, you must run the Signal Path Compensation (SPC) procedure in the Utilities menu after a 20-minute warm-up period. This returns the instrument solid state drive to the initial state shipped from the factory.

Scrubbing the drive will not affect calibration of the instrument, since the factory calibration constants are stored on the acquisition board, entirely separate from any acquisition data or user files. You can completely erase or remove any secure data without affecting the calibration of the instrument. You can also calibrate the instrument in a nonsecure site, and then use the instrument in a secure area without recalibration.

Powering on Windows for the first time

The instrument goes through a series of configuration boot-ups 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 33)).

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 the instructions below to verify the Windows 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 the instructions below to verify the Windows 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 the instructions below to verify the Windows 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 the instructions below. If Windows lost activation when installed on the current instrument, follow the instructions above 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 need to have any active maintenance license (subscription, perpetual, or floating license within its term) to update the TekScope application. 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 TekScope 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:

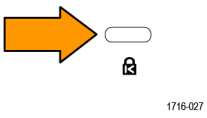
Follow the above instructions on a network-connected PC,

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 instrument

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

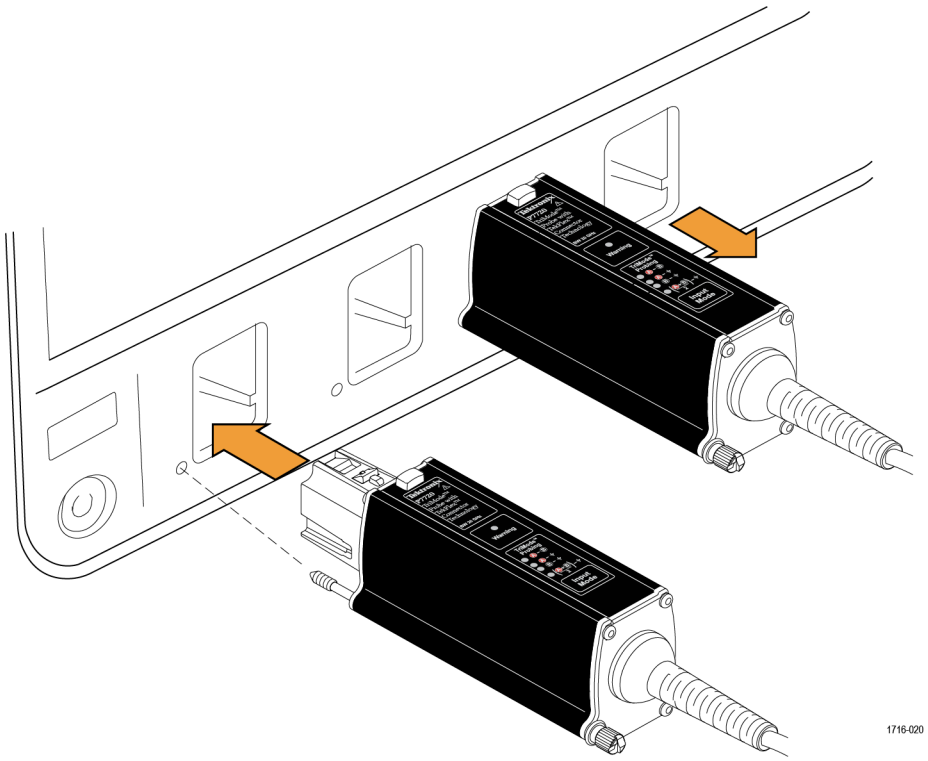
To secure the instrument to a workbench, rack, or other location, attach a standard laptop security lock to the rear panel of the instrument.



1716-027

Connecting probes to the instrument

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



Connecting probes to the 7 Series DPO

Connect the probe to instrument

Connect supported Tektronix probes and adaptors by pushing them into a TekConnect connector. The probe base latch locks with a 'click' when the probe is fully seated.

TekConnect probes automatically set the channel input parameters for that probe (bandwidth, attenuation, termination, and so on).

Disconnect the probe

To disconnect, turn the thumbscrew counter-clockwise. Press the latch release button and pull the probe away from the instrument.

NOTE: Connecting a probe or adaptor 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).

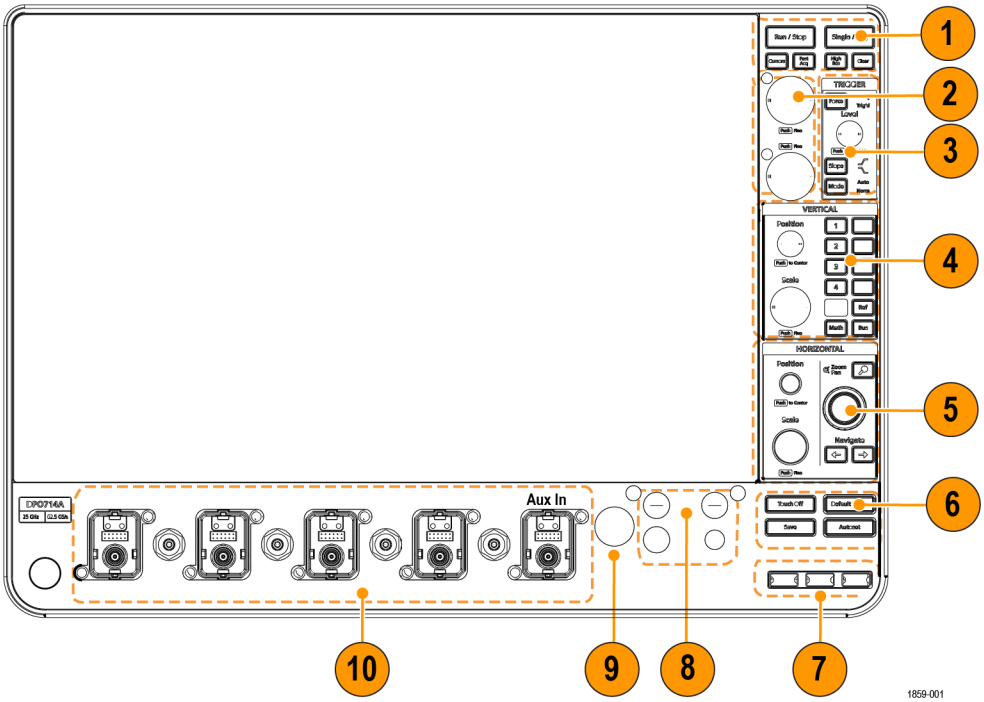
Rackmount option information

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

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

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.



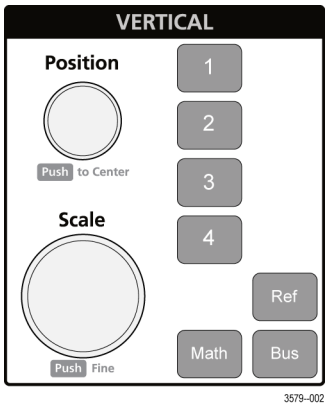
7 Series DPO controls

1. **Acquisition** and **Cursors** controls:
   - **Run/Stop** starts and stops waveform acquisition. The button color indicates the acquisition status (green = running and acquiring; red = stopped). When stopped, the oscilloscope shows waveforms from the last completed acquisition. The Run/Stop button on the screen also shows the acquisition status.
   - **Cursors** button turns screen cursors on or off. Use the Multipurpose knobs to move the cursors. Double-tap the cursor readouts, or on a cursor bar (line), to open the configuration menu to set cursor types and functionality.
   - **FastAcq™** 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. This information can be used to construct a trigger to capture these infrequent events.
   - **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.
   - **Clear** deletes the current acquisitions and measurement values from memory.
2. **Multipurpose knobs**: The Multipurpose knobs A and B move cursors and set parameter values in configuration menu input fields. Selecting an menu field that can use a Multipurpose knob assigns the indicated knob to change the value in that input field. The ring around each knob lights when you can use that knob to do an action. Push a Multipurpose knob to enable the **Fine** mode for making smaller increment changes. Push the knob again to exit **Fine** mode.
3. **Trigger** controls:
   - **Force** forces a trigger event at a random point in the waveform and captures the acquisition.
   - **Level** sets the amplitude level that the signal must pass through to be considered a valid transition. The color of the **Level** knob LED 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:

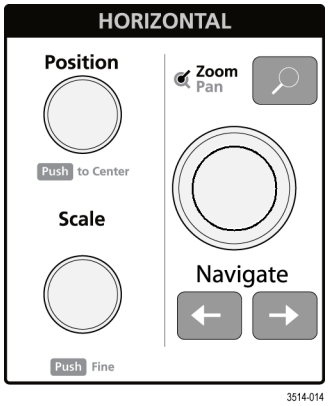


- **Position** moves the selected waveform (Channel, Math, Reference, Bus) and its graticule up or down on the screen. The color of the **Position** knob indicates which waveform the knob is controlling. Push the knob to set the threshold level to 50% of the peak-to-peak amplitude range of the signal.
- **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 Channel, Math, Reference, or Bus waveforms. The number of channel buttons depends on the instrument model. The buttons operate 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).
- **Math** button adds or selects a Math waveform on the Waveform view.
  - If no Math waveform exists, pushing the **Math** button adds a Math waveform to the Waveform view and opens the Math configuration menu.
  - If only one Math waveform is displayed, pushing the button turns off the Math waveform (removes it from Waveform view). Push the button again to display the waveform.
  - If two or more Math waveforms are displayed, pushing the button cycles through selecting each math waveform.
- **Ref** button adds or selects a Reference (saved) waveform on the Waveform view.

- 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.

- **Bus** button adds or selects a bus waveform on the Waveform view.
  - 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.

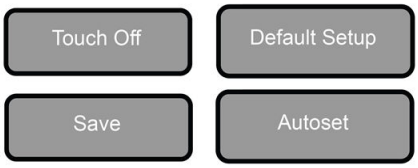
5. **Horizontal** controls:



- **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 close zoom mode.See .
- **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 46).

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

6. **Touch Off**, **Default Setup**, **Save** and **Autoset** buttons



- **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.
  - 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 .

7. **USB Host ports** (USB 3.0): 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.
8. **Differential Fast Edge** differential fast edge step output.

   Ground Antistatic Connection banana type ground connection. Discharge the static voltage from your body by wearing a grounded antistatic wrist strap while connecting and disconnecting cables and TekConnect adapters.
9. DC Probe Cal. See Compensate the TPP Series probesTriMode probe DC calibration (on page 65).
10. **DPO714AX/DPO718AX:** TekConnect input channel probe connectors (4) and TekConnect AUX In connector (1); **DPO718AX only:** 2.92 mm Planar Crown® input channels (4).

    TekConnect channels have up to 25 GH bandwidth and 125 GS/s sample rate. TekConnect channels offer multiple connection options with a wide range of P7000 Series Performance Probes and TCA adapters.

    Aux In is an auxiliary trigger input connector.

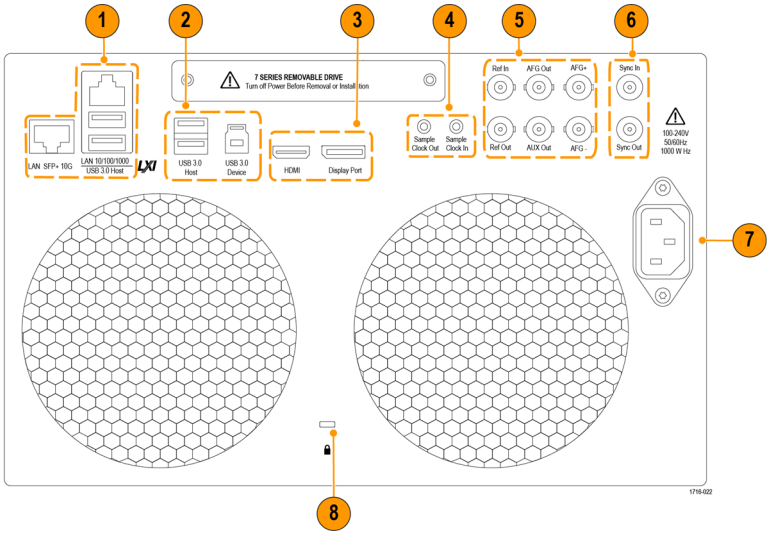
    If you enable one or more 2.92 mm channels, then the sample rate for all TekConnect channels is reduced to 62.5 GS/s. The sample rate is 125 GS/s on all four TekConnect channels, or 62.5 GS/s on all TekConnect and 2.92 mm channels if one or more 2.92 mm channels is enabled.

CAUTION: IHand tighten Planar Crown® connectors. Tool tightening may result in damage to this connector type.

Rear panel connections

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

1.



1. **LAN SFP+ 10G** is a flexible 10 Gb Ethernet port module that allows you to connect a variety of ways.

   LAN connector (RJ-45) connects the instrument to a 10/100/1000 Base-T local area network.

2. **USB 3.0 Host** ports let you connect a USB memory device, keyboard, or mouse.

   USB 3.0 Device port lets you connect to a PC to remotely control the instrument using USBTMC protocol.

3. **HDMI** and **Display Port** lets you connect an external monitor or projector to show the instrument's graphical user interface.

4. **Sample Clock In** lets you connect the 7.8125 GHz clock signal to oscilloscope.

   Sample Clock Out outputs a 7.8125 GHz clock signal from the oscilloscope.

5. **Ref In** and **Ref Out** lets you connect a high-precision 10 MHz reference signal to the oscilloscope for more accurate measurements.

   AUX Out generates a signal transition on a trigger event, outputs a 10 MHz reference signal, or outputs a synchronization signal from the AFG.

   AFG Out is the signal output for the optional Arbitrary Function Generator (AFG) feature.

   AFG + and **AFG -** are differential output connectors.

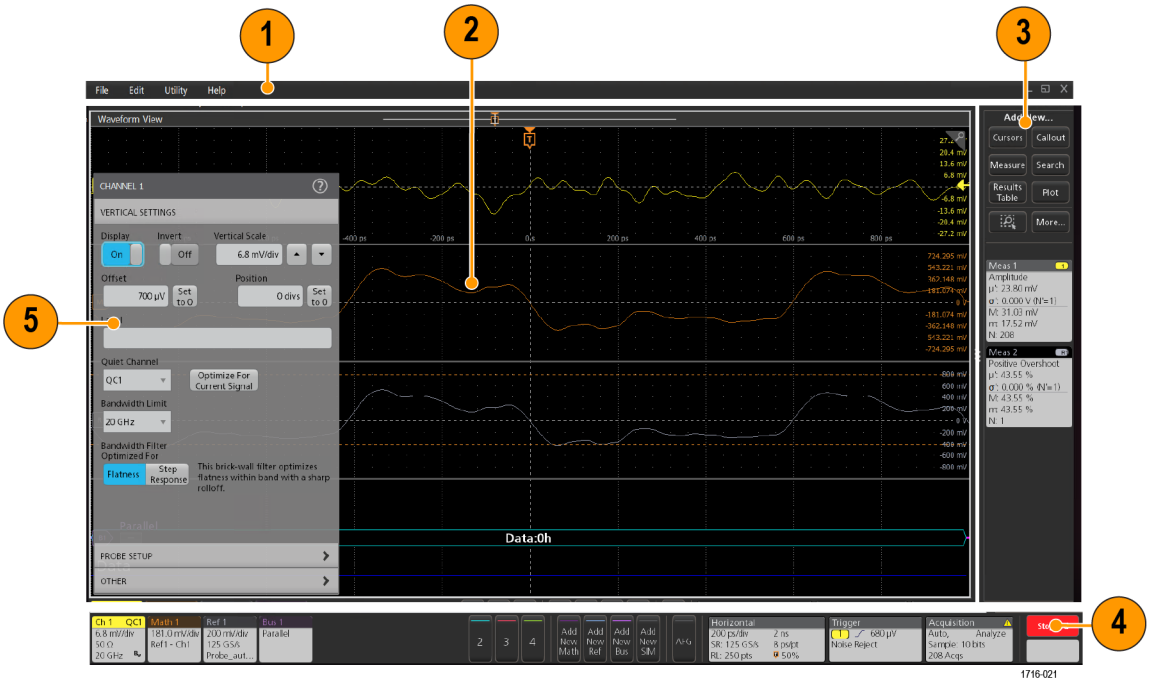
6. **Sync In** and **Sync Out** provides a synchronization signal to external devices.

7. **Power cord** connector. Use only the power cord specified for this product and certified for the country of use.

8. **Security** lock connector lets you use a standard PC/laptop lock cable to secure the instrument to a work bench or other location.

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 User interface elements (on page 43).

   You can also add Histogram, Spectral, 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 callout, plots, and result tables to the screen, and add measurements to the Results bar. To remove a measurement, search or other badge from the Results Bar simply flick it off screen. The controls are:
   - The **Cursors** button displays on-screen cursors in the selected view. Touch and drag, or use the Multipurpose knobs, to move the cursors. Double-tap on a cursor, or on the cursor readouts, to open a configuration menu to set cursor types and related functions.
   - The **Measure** button opens a configuration menu from which to select and add measurements to the Results bar. Each measurement you add has a separate badge. Double-tap a measurement badge to open its configuration menu.
   - The **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.
   - To remove a measurement, search or other badge from the Results Bar simply flick it off screen.

- The **Callout** button adds a Callout object to the selected view. Double-tap the Callout text to open a configuration menu to change the type of callout, text and font characteristics. Drag any callout other than bookmark to any location on the oscilloscope screen view. Bookmarks callout can only be added to waveviews and spectrum views.
- 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, Eye Diagram, or Transfer Function 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 46). See Add a measurement (on page 80). See Add a Search (on page 86).
- The **Zoom icon** button at the up right of the **Results Bar** lets you to draw a box on the screen to zoom in on an area of interest, drawing segments for mask testing, or drawing areas to define visual trigger conditions.
- The **More...** button at the up right of the **Results Bar** allows you to select Zoom, Visual trigger, Mask, or Waveform Histogram.

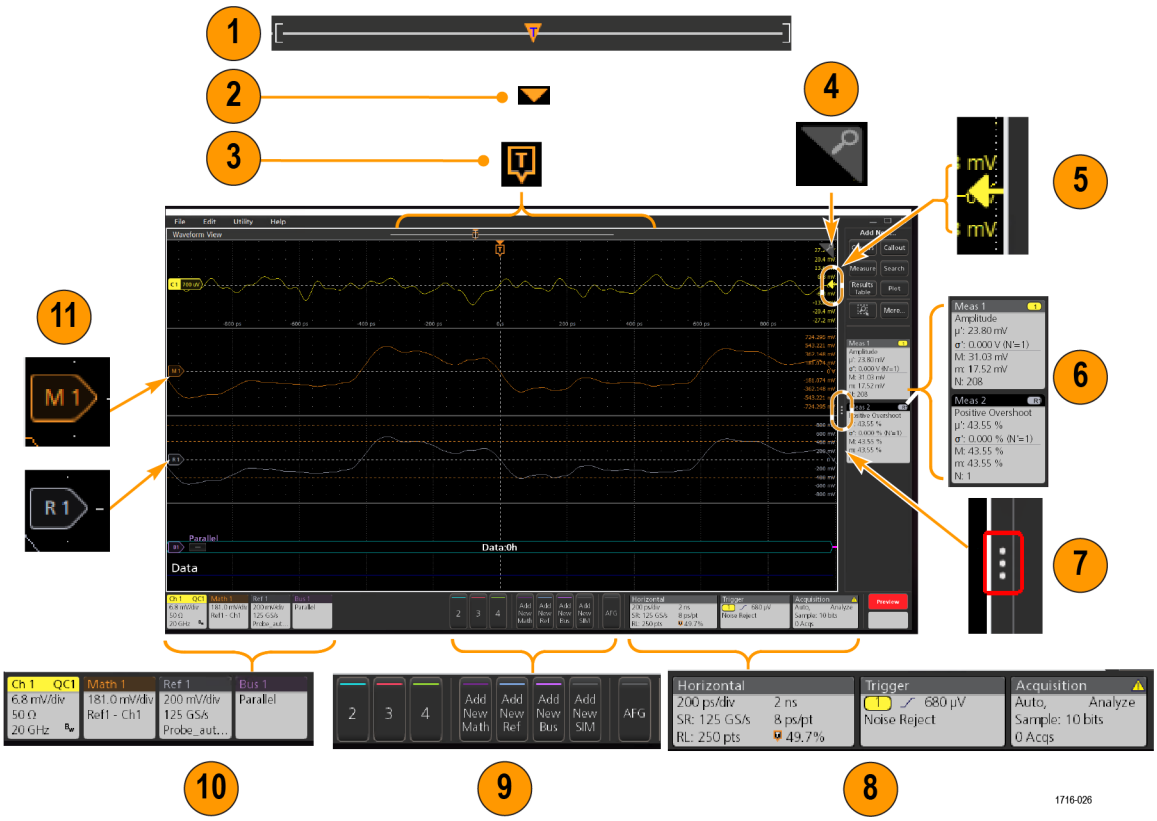
4. The **Settings Bar** contains System badges for setting Horizontal, Trigger, Acquisition, and Date/Time parameters; **Inactive Channel** buttons to turn on channels; **Add New Waveform** buttons to add math, reference, and bus waveforms to the display; and Channel and Waveform badges that let you 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 46).

   The SIM option enables **Add New SIM** button that lets you to create and configure a signal integrity model for de-embedding, embedding the simulation model.

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 56).

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 Zoom user interface (on page 58).

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 icons show 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 46). See Add a measurement (on page 80).
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 46).
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 **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.

   If the SIM option is installed, select the **Add New SIM** button to create and configure a signal integrity model for de-embedding, embedding the simulation model.

10. Double-tap a badge to open its associated configuration menu. See Badges (on page 46). See Configuration menus (on page 56).

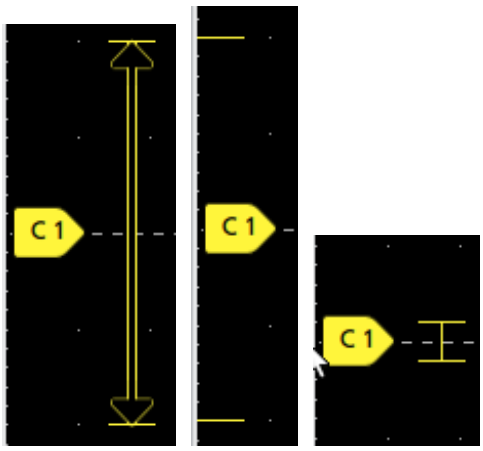
    If you add more Channel or Waveform badges than can fit in the waveform badge display area, tap the scroll buttons at each end of the waveform badge area to scroll and display hidden badges.

11. The Waveform Handles on each waveform identify the source of that waveform (C*x* for channels, M*x* for Math waveforms, R*x* for Reference waveforms, B*x* for bus waveforms). The waveform handles are at the zero-volt level of the waveform 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.

    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.

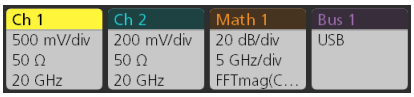


Badges

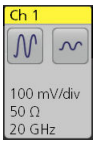
Badges are rectangular icons that show waveform, measurement, and instrument settings or readouts. Badges also provide fast access to configuration menus. The badge types are Channel, Waveform, Measurement, Search, and System.

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 and open the badge right-click menu to access a quick-action menu.

Right click menu for Channel and Waveform badges

Badge readout	Description
Turn Ch On/Off	Tap to turn On or Off the channel depending on the state.
Copy to Ref Wfm	Select Copy to Ref Wfm to save the waveform and recall it as a new reference waveform. This option is unavailable for Bus reference waveforms and SIM reference waveforms.
Configure Ch	Tap to configure instrument channel. This opens the vertical setting panel.
Termination	Set the termination value for channel waveform.
Coupling	Set the coupling value for channel waveform.

Badge readout	Description
Bandwidth	Set the bandwidth for the channel waveform.
Label	Enter label text for the channel using virtual keypad or keypad connected to instrument.

There are two ways to delete Channel and Waveform badges.

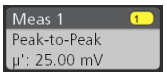
- Right-click the badge and turn it off.
- Flick the badge off the bottom edge of the display to remove it from the **Settings** bar. Flicking upwards from the bottom edge of the **Settings** bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

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.

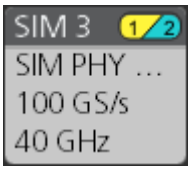


Double-tap a Measurement badge to open its configuration menu to change or refine settings. The default measurement badge readout shows the measurement's mean (μ) value.

SIM badges

SIM badges are shown in the settings bar, located along the bottom left of the screen. Each simulation model has its own badge. The badge shows high-level settings for each model. Double-tap a badge to open its configuration menu.

SIM badges for the simulation (Sim 1, Sim 2, etc.) and shows the channel chicklet(s) of the sources that the simulation is based on.



Badge readout	Description
SIM X	A label on the top of badge defines the simulation model name. Example: SIM 1, SIM 2. The the top right corner of the badge displays the source channel numbers used for simulation.
Model	A label to SIM model for user reference.
Sample rate	Displays the sample rate at the model is designed to operate at.
Bandwidth limit	Displays the global bandwidth limit.

There are two ways to delete SIM badges.

- Right-click the badge and turn it off.
- Flick the badge off the bottom edge of the display to remove it from the Settings bar. Flicking upwards from the bottom edge of the Settings bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

SIM badges are listed in the SIM order unless you have moved them. It may also display short error or warning messages.

Mask Test badge

The mask test results and measurement statistics are displayed in the **Mask Test** badge in the Results bar. The badge is created when the first segment of a mask is defined.



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

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

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

There are two ways to delete Channel and Waveform badges.

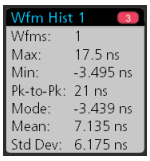
- Right-click the badge and turn it off.
- Flick the badge off the right edge of the display to remove it from the **Results** bar. Flicking left from the right edge of the **Results** bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

See for the mask test configuration settings Create a Mask (on page 114) for more information masks and Mask Test badge configuration menu (on page 290).

Waveform Histogram badges

The Waveform Histogram badges are located in the **Results** bar. The badge title shows the histogram source.

To add the Waveform Histogram badge to Results bar, set the **Display** to **On** from the Result Badge menu. Double-tap a Waveform Histogram badge to open its configuration menu to change settings.



The histogram badge displays the measurements which are checked in the Result Badge menu. To add more histogram measurements to the badge refer Waveform Histogram Result Badge menu (on page 535).

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

There are two ways to delete Waveform Histogram badges.

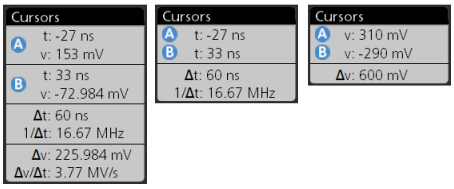
- Right-click the badge and select **Delete Histogram**.
- Flick the badge off the right edge of the display to remove it from the Results bar. Flicking left from the right edge of the Results bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

See also

- Create a waveform histogram (on page 149)
- Waveform Histogram configuration menu (on page 533)

Cursor badges

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



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

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

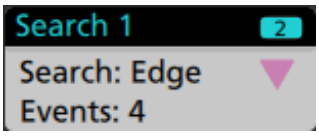
You can drag the badge to change its position in the **Results** bar and open the badge right-click menu to access a quick-action menu.

There are two ways to delete Channel and Waveform badges.

- Right-click the badge and turn it off.
- Flick the badge off the right edge of the display to remove it from the **Results** bar. Flicking left from the right edge of the **Results** bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

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 and open the badge right-click menu to access a quick-action menu.

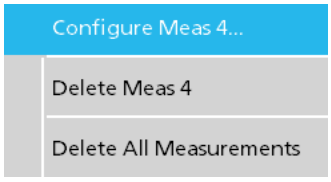
There are two ways to delete Channel and Waveform badges.

- Right-click the badge and turn it off.
- Flick the badge off the right edge of the display to remove it from the **Results** bar. Flicking left from the right edge of the **Results** bar recovers the badge. Badge recovery is only possible within 10 seconds of removal.

Deleting mass Measurements/Searches badges at once

This badge helps you to delete/remove many number measurements or searches, which are in the Results bar.

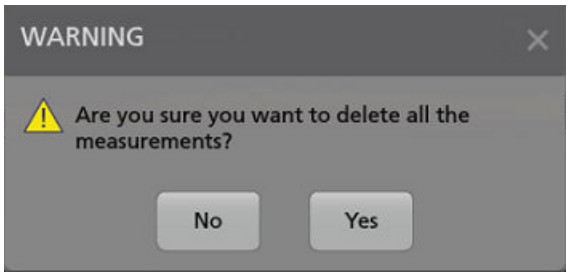
1. Select and right click the Measurement/Search badge in the **Results** bar, which displays the dialog box as shown:



Controls	Description
Configure Measurement/Search	Configure Measurement or Search badges

Controls	Description
Delete Measurement/Search/Histogram	Deletes the selected Measurement (standard, jitter, etc.)/Search/Histogram badge in the Results bar.
Delete All Measurement/Search/ Histogram	Deletes all the Measurement (standard, jitter, etc.)/Search/Histogram badges in the Results bar.

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

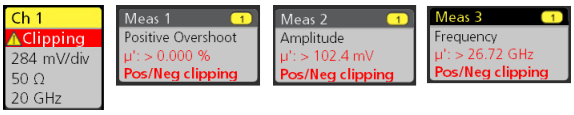


3. The dialog box provides you a checkbox that gives the choice to bypass remaining information dialogs.
   - **Don't ask for remaining items**: Default is unchecked. If you leave it unchecked and clear the information dialog, then the dialog will appear again for the next measurement deletion.
   - If the box is checked, it proceeds with deleting the rest of the items without bringing up the dialog box again. The dialog box appears for each set of measurements you want to delete.

Signal clipping and badges

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

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

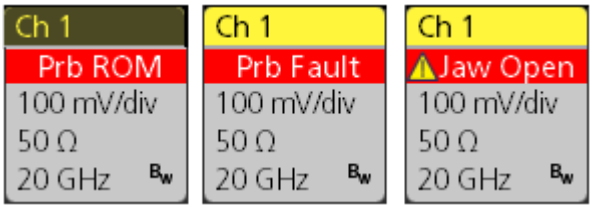


To close the clipping message, change the vertical scale to show the entire waveform, disconnect the probe tip from the excessive voltage source, and check that you are probing the correct signal using the correct probe.

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

Error messages and badges

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

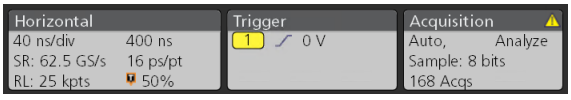


To remove the message from the badge, clear the error as indicated in the table.

Error message	Description
AutoZero	Autozero failed.
Clipping	Input signal is clipped. Change the probe attenuation factor or adjust vertical scale and re-run.
Degauss	Degauss current probes by selecting the Degauss/Autozero button on each probe and re-run.
Jaw Open	Ensure the jaw is closed on all current probes and re-run.
Over Rng	The signal voltage or current is over range. Reduce the signal amplitude.
Prb Com	Probe communication timed out. Remove and re-attach the probe.
Prb Disab	Probe disabled. Remove and re-attach the probe. If error persists, contact Tektronix Service.
Prb Fault	Critical probe fault. Remove and re-attach the probe. If the problem persists, contact Tektronix service.
Prb ROM	Unable to read probe ROM. Remove and re-attach the probe.
SelfCal	Probe self calibration failed.
Temp	The probe has experienced an over temperature condition. Remove the probe from the high temperature area.
Tip Fault	The probe tip has a fault. Remove and replace the probe tip.

System badges

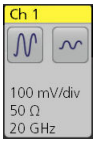
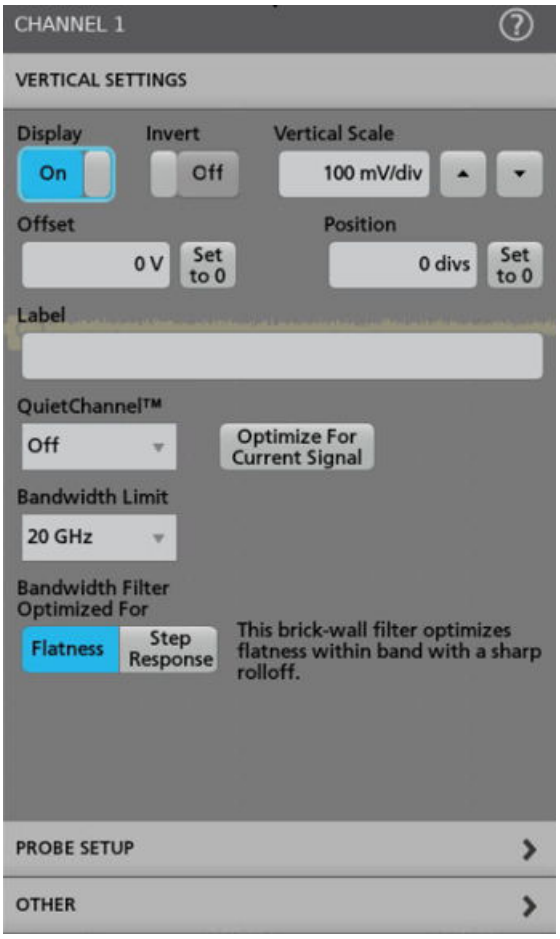
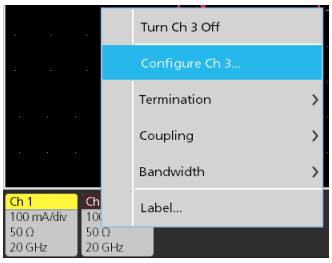
System badges (in the settings bar) display the main Horizontal, Acquisition, and Trigger settings. You cannot delete System badges.



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

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

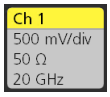
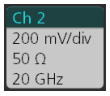
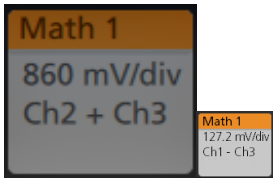
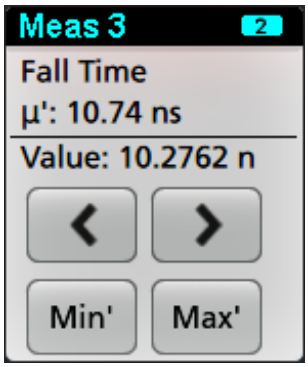
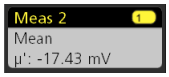
Common badge actions

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

Action	Result	Example
Flick	Flick the badge off the bottom edge of the display to remove it from the **Settings** bar. Flick the badge off the right edge of the display to remove it from the **Results** bar. Flick from the right or bottom edge to recover a removed badge. This action is only possible within 10 seconds of badge removal.	

Badge selection status

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

Badge type	Selected	Unselected	Turned off or in use
Channel or Waveform			
Measurement			N/A

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.

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 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

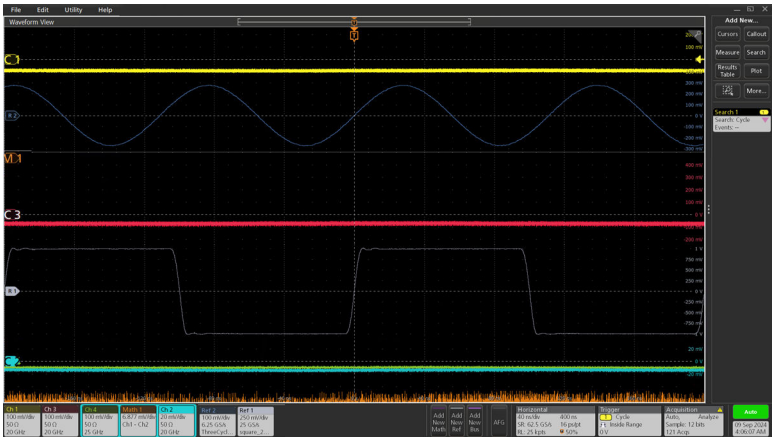
- A Measurement or Search badge can only be moved within the Results Bar.
- Dragging a badge to a new location causes the non-selected badges to move to create the position at which to insert the badge.
- Once you have moved any Results badge to a new position, adding a new Measurement or Search badge 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

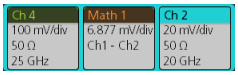
You can group signal badges in the Settings bar to display multiple waveforms in a single slice. See Group signal badges in the Settings bar (on page 56) for more information.

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.

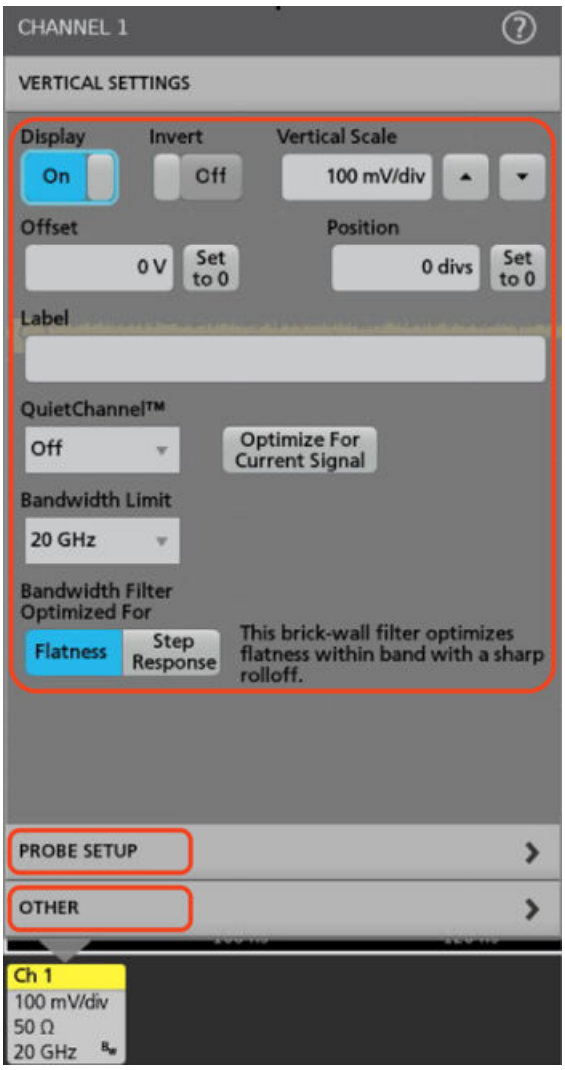
Badge group caveats

- 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

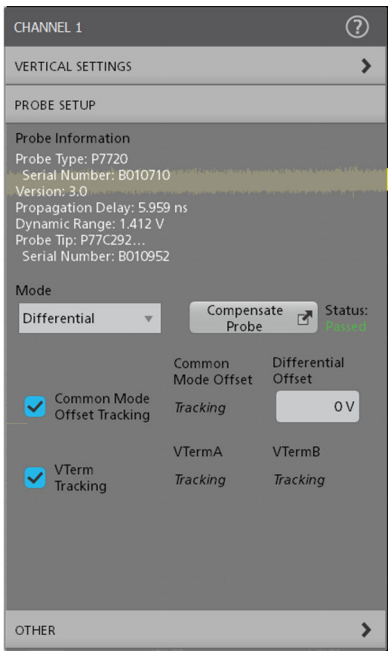
Configuration menus let you quickly set the parameters for channels, system settings (Horizontal, Trigger, Acquisition), measurements, cursor readouts, Waveform and Plot views, callout text, and so on.

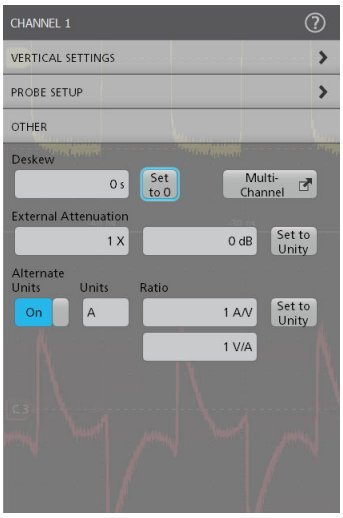
Double-tap an item (badge, **Waveform View** or **Plot View**, cursor readouts, callout text, and so on) to open its configuration menu. For example, double-tap a Channel badge in the **Settings Bar** to open its configuration menu.



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

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



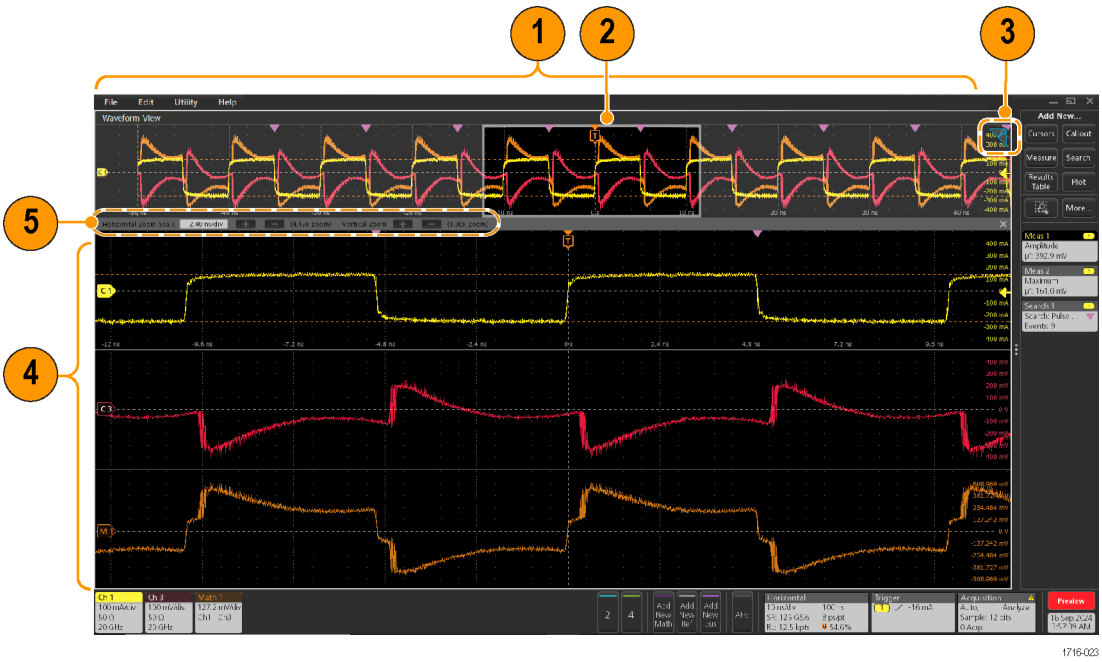


Tap anywhere outside a configuration menu to close it.

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

Zoom user interface

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



1. The **Zoom Overview** shows the entire waveform record. All waveforms are shown in Overlay mode in the Zoom Overview area. Using pinch and expand gestures on the Zoom Overview waveforms changes the horizontal time base settings.
2. The **Zoom Box** shows the area of the Zoom Overview to display in the Zoom View (see 5). You can touch and drag the box to move the area to view. Moving the Zoom Box, or changing its position, does not change the horizontal time base settings. You can also use the zoom **Pan** knob to move the Zoom Box left or right.
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), drawing areas for the **Visual Trigger** function, drawing segments for **Mask** testing, and drawing the **Waveform Histogram** for waveform analysis. 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, **Visual Trigger** mode, and **Mask** mode, double-tap the **DRAW-A-BOX** button and select one of the options. Search for the **Visual Trigger** and **Mask Testing** topics in the oscilloscope Help for more information.
5. The **Zoom View** shows the zoomed waveforms, as marked by the Zoom Box, in the Zoom Waveform Record View. Use pinch and/or drag options in the zoom view to change the zoomed area of interest. Pinch, expand, and drag gestures in the Zoom View only change zoom magnification settings and Zoom Box position.
6. Use the **Zoom Title Bar** controls to adjust the vertical and horizontal size of the zoom area. Click or tap the + or - buttons or use the A and B multipurpose knobs.

   In a waveform view, if both Cursor and Zoom are turned on, use the **Zoom Box** and **Cursors** button to change the multipurpose knob functionality. Tap the **Zoom Title Bar** to assign the knobs to adjust the zoom or tap the **Cursors** button to assign knobs to adjust the cursors.

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

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

Using the touch screen interface for common tasks

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

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

Task	Touchscreen UI action	Mouse action
Add a channel, math, reference, or bus waveform to the screen.	Tap an inactive channel button, **Add New Math**, **Add New Reference**, or **Add New Bus** button.	Click an inactive channel button, **Add New Math**, **Add New Reference**, or **Add New Bus** button.
Select a channel, math, reference, or bus waveform to make it active	Stacked or Overlay mode: Tap the Channel or Waveform badge.	Stacked or Overlay mode: Left-click the Channel or Waveform badge.
	Stacked mode: Tap the channel, math, reference, or bus waveform slice or handle.	Stacked mode: Left-click the channel, math, reference, or bus waveform slice or handle.
	Overlay mode: Tap the channel or waveform handle.	Overlay mode: Left-click the channel or waveform handle.
Display scale or navigation buttons on a badge (waveform, measurement, search, horizontal). Not all measurement or search badges display navigation buttons.	Tap the badge.	Click the badge.
Open a configuration menu on any item (all badges, views, cursor readouts, labels, and so on).	Double-tap the badge, view, or other object.	Double-click the badge, view, or other object.
Open a right-click menu (badges, views).	Touch and hold on the badge, Waveform View, Plot view, or other screen item until a menu opens.	Right-click the object.
Close a configuration menu. Some dialog boxes will not close until you click an OK, Close, or other button in the dialog.	Tap anywhere outside the menu or dialog.	Click anywhere outside the menu or dialog.
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 callout. Callouts are screen objects and are not associated with any particular waveform channel or slice.	Touch and hold on a callout and quickly start to drag, then move to new position. Start to move the callout as soon as selected (highlighted), otherwise the UI opens the right-click menu.	Click and hold the right mouse button on the callout and quickly start to drag, then move to the new position.

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

Accessing application help

Use the instrument 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.

NOTE: The Help tool does not have a virtual keyboard. Connect a keyboard to your instrument to enter text in the search fields of the Search or Index tabs.

From the help browser, select one of the following tabs:

- **Contents Tab**. Click any entry to display information on the subject.
- **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 click Display to open the topic.

Other help features

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

ESD Prevention Guidelines

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

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

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

Cleaning, service, maintenance, and packing

Cleaning

Use a dry, soft cotton cloth to clean the outside of the unit. If any dirt remains, use a cloth or swab dipped in a 70% isopropyl alcohol solution. Use a swab to clean narrow spaces around controls and connectors. Do not use any liquid cleaning agents or chemicals that could damage the touch screen, case, controls, markings or labels, or possibly infiltrate the case.

Service, maintenance, and packing

See the *7 Series DPO Performance Oscilloscope Service Manual* for additional information about packing your instrument for shipping in the HC7B transit case, service, and maintenance procedures. You can also refer to the *HC7B Transit Case Instructions* for shipping information.

Configure the instrument

Configurations to help operate your instrument efficiently.

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).

About this task

Procedure



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

   To turn on date/time display again, double-tap in the blank area where the date/time badge was displayed to open the configuration menu, and set the **Display** button to **On**.
3. Select a time format (**12 Hour** or **24 Hour**).
4. Tap the **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.

1. Power on the oscilloscope.
2. Tap **Utility** > **Self test**. Check that all tests listed show **Pass**.
3. Connect a cable between one of the outputs of the Differential Fast Edge output and Channel 1.
4. Ensure that a termination cap is secured to the other output of the Differential Fast Edge.
5. Push the **Autoset** button. You should see a square wave in the display (approximately 600 mV_{P-P})

Download and install the latest instrument firmware

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

In order to install a new version of instrument firmware, any current maintenance period must be active. There are three ways to have maintenance:

- Active subscription
- Active perpetural license with a maintenance term
- Active mainframe license

Save any important on-instrument files (waveforms, screen captures, 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.

1. Power on the oscilloscope.
2. Determine the current version of firmware installed on the instrument in the **About** window under the **Help** menu
3. Check the version of the latest available firmware at the tek.com/software. Enter the model number of your oscilloscope in the search box and search for the firmware. If the firmware version on the web is newer than what is currently installed, update your instrument firmware.
4. Install the firmware on embedded OS instruments.
   a. Download the latest firmware and unzip the files on your PC.
   b. Copy the designated firmware file into the root folder of a USB flash drive.
   c. Insert the USB flash drive into a USB port on your oscilloscope. The instrument automatically recognizes the firmware install file and starts the installation process.
   d. Follow any on-screen instructions. The oscilloscope take about 10 minutes to install the new firmware. Do not remove the USB flash drive or power off the instrument during this time.
   e. After the oscilloscope displays a message that it has completed the upgrade, the scope will reboot automatically.

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

Run Signal Path Compensation (SPC)

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

About this task

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

Before you begin

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

Procedure

1. Power on and warm up the instrument for at least 20 minutes.
2. Tap **Utility > Calibration**.
3. Tap **Run SPC**. The **SPC Status** readout shows **Running** while SPC is running. SPC can take 15 minutes approximately per channel to run, so wait until the SPC Status message changes to **Pass** before reconnecting probes or cables and using the instrument.

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

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

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

TriMode probe DC calibration

After you perform a functional check of the probe, run a probe calibration routine on each channel that you use. The probe calibration operation minimizes measurement errors by optimizing the DC gain and offset of the probe. Individual calibration constants are stored for all TriMode settings, on each probe, on each channel.

About this task

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.

NOTE: To avoid ESD damage to the probe, always use an antistatic wrist strap (provided with your probe), and work at a static-approved workstation when you handle the probe.

- 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.

Before you begin

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

Procedure

Use this procedure to compensate supported TriMode probes that show a **Default** status when connected to the oscilloscope.

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

1. Connect a supported probe to an input channel. Refer to your specific probe's manual for setup instructions for TriMode Probe DC Calibration.
2. Double-tap the badge of the channel that you want to compensate.
3. 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.

4. Tap **Compensate Probe** to open the **Probe Compensation** dialog.
5. Tap **Compensate Probe** to run the probe compensation.
6. The probe compensation is finished when the Probe Compensation Status displays **Pass**. Disconnect the probe from the setup.
7. Repeat these steps for each supported passive probe that you want to compensate for this channel.
8. 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.

Connect to a network (LAN)

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

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

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

You can use the LAN SFP+ port to connect the instrument to your network. This port is a flexible 10 Gb Ethernet port that allows you to connect a variety of ways. The SFP+ port can be used with:

- **10GBASE-T SFP+ to RJ-45 modules**: Enables connections to standard RJ-45 Ethernet networks using CAT6 or higher Ethernet cables.
- **10GBASE-SR SFP+ optical modules**: Enables connections to optical Ethernet networks using fiber patch cables.
- **10GBASE-CU SFP+ direct attach cables**: Allows for direct electrical connections between SFP+ ports using inexpensive passive cables.

The SFP+ port is hot-swappable, so it can be swapped out while the instrument and the switch are turned on.

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 the LAN SFP+ port on the instrument to your network via one of the connection methods listed above.
2. Select **Utility > I/O** on the menu bar to open the I/O configuration menu.
3. Tap **LAN SFP+ PORT**.
4. 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.
5. Tap **Test Connection** to verify that the network connection is working. The **LAN SFP+ 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.

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 Connect to a network (LAN) (on page 66).

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 Drive** menu.
3. Select **Network** in the Drive Type.
4. Select the drive letter to assign to the network drive from the **Drive Letter** list.
5. Tap **Name** or **IP** to specify how you will enter the host name (server) of the network mount location or PC.
6. 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.
7. 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.)
8. If access to this network location is controlled, enter the required information in the **User Name**, **Password**, and **Domain/Workgroup** fields.
9. 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.

CAUTION: The Linux ScopeApp does not support SMB version 1.0 for CIFS network mounts.

Mount a network drive on the 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 Connect to a network (LAN) (on page 66).

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 file utilities window.
2. Select **Network** in the Drive Type.
3. Click on **Open File Explorer** to open the windows explorer window.
4. Tap **This PC** in the path field to move to the This PC directory.
5. Tap and hold the **This PC** text on the left side of the pane to open the context menu.
6. Tap **Map Network Drive** to open the **Map Network Drive** menu.
7. 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.
8. 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.

Mount TekDrive

TekDrive is a collaborative T&M data workspace provided with the purchase a supported Tektronix product.

Before you begin

This mounting procedure is applicable for both Windows and Linux platforms. You must have internet connectivity access to perform this procedure.

Browse to http://drive.tekcloud.com, register, and create a new account.

- The TekDrive mount feature is supported on 7 Series DPO instruments.
- The maximum upload file size limit to TekDrive is restricted to 25 GB.
- When the TekDrive is enabled, you cannot copy and paste for files and folders in the TekDrive.
- In some scenarios, recalling the files from the TekDrive is not supported and you will not see it listed in the file browser, but you can continue recalling the file through USB/Network or Local drives. For example, Math Filter files cannot be recalled/applied from the Math advanced menu if it is stored in the TekDrive.
- When a TekDrive is unmounted or auto-disconnected, the waveform files activated from it would continue to work until you recall the defaults, run TekSecure®, or change the reference waveform file.

About this task

Use TekDrive to upload, store, organize, search, download, and share arbitrary files from any connected device. Use the following procedure to mount (map) a TekDrive.

Procedure

1. Tap **File > File Utilities**.
2. Tap **Mount** to open the **Mount Drive** menu.
3. Select the **TekDrive** button under **Drive Type**
4. Enter the **Drive Name**.
5. Select the required **Auto Disconnect** option.
6. Tap **Connect to TekDrive** to display the window with the QR and numeric code for TekDrive activation.
7. Browse to drive.tekcloud.com/activate to login using your credentials and enter the numeric code displayed in the Mount Drive window.

   Register and create a new account if you are a new user.

   A message is displayed when activation is successful.
8. After activation, click **OK, mount Tekdrive** to complete the mounting process.

   The connected TekDrive is available in all file browsing windows.

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 the 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. Open Windows Explorer window.
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.

Unmount a TekDrive

Use this procedure to unmount (remove, disconnect) a TekDrive.

Procedure

1. Tap **File > File Utilities**.
2. Select the TekDrive to unmount in the **Drive** column.
3. Tap **Unmount**.

   The drive is unmounted and it is removed from the **Drive** column.

Connect the oscilloscope to a PC using a USB cable

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

1. On the oscilloscope, select **Utility > I/O** from the menu bar.
2. Tap **USB Device Port Settings**.
3. Confirm that the USB Device Port control is **On** (default setting).
4. Connect a USB cable from the PC to the USB **Device** port on the instrument.
5. If using the USB connection to remotely control the oscilloscope using GPIB commands, set the **GPIB Talk/Listen Address** for your configuration (0 to 30).

Remote access from a Web browser

You can remotely access your network-connected standard instrument (not running Windows) 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 the *Remote access to a Windows 10 instrument* (on page 71) topic in the Help.

Prerequisites:

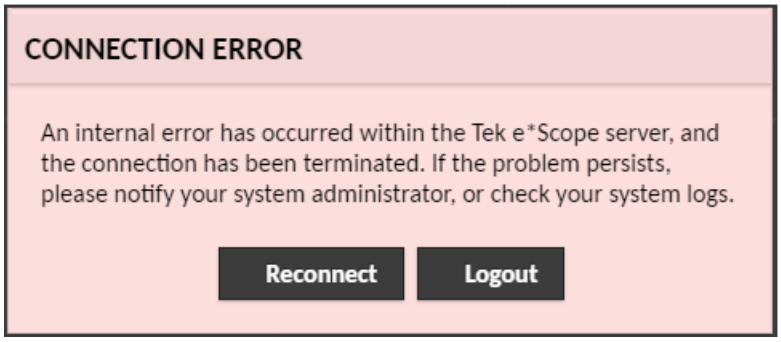
- The instrument must be connected to, and accessible from, the network to which the PC is connected. See Connect to a network (LAN) (on page 66)
- The IP address of the instrument that you want to access. To determine the instrument's IP address, select **Utility > IO** in the instrument menu bar and view the network settings in the **LAN** panel.
- You are accessing a standard (instrument that does not have the Windows OS option installed).

Use below procedure to remotely access the UI controls and screen for standard (non Windows 10) instruments using e*Scope® functionality.

1. Open a Web browser on a PC connected to the same network as the instrument.
2. Enter just the instrument 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 instrument.
3. Select **Instrument Control (e*Scope)**. The browser displays the instrument screen. If a password is set for the e*Scope, an Enter Password window will popup. Enter the password and tap **Submit**. If the password is input incorrectly five times, it will block the e*Scope for five minutes.
4. Use a mouse to select and interact with the instrument controls shown in the Web browser. If your remote PC or laptop has a touch screen monitor, you can use the remote touchscreen monitor to access the instrument controls.

   Close the browser to logout the session.

NOTE: If connection error occurs, refresh the page in the web browser or restart the web browser for seamless interaction with the instrument remotely.



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:

1. Open a connection to the instrument using **e*Scope**.
2. Select and copy the text on your PC.
3. In e*Scope, press **Ctrl-Alt-Shift** to open the **Clipboard** menu.
4. Paste the text into the Clipboard field.
5. Press **Ctrl-Alt-Shift** to close the browser Clipboard menu.
6. 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.
7. Press **Ctrl-V** (on real keyboard or from virtual keyboard) to paste the text from the e*Scope browser clipboard to the menu field.
8. Repeat steps 2 to 7 above 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 66)
- 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 72) and Install TightVNC on a PC (on page 72).

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.
   b. Under the **Download TightVNC for Windows (Version 2.8.8)** heading, click **Installer for Windows (64 bit)**.
   c. 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 71).

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.
   b. Under the **Download TightVNC for Windows (Version 2.8.8)** heading, click **Installer for Windows (64 bit)**.
   c. 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 71).

Deskew analog input channels

Critical Timing measurements on multiple channels require that all probes be adjusted, or deskewed, to compensate for signal timing differences between probes.

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.

Measurement method

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

1. Connect all probes that you want to deskew to the oscilloscope.
2. Use the appropriate deskew fixture for the probe to connect to a reference signal.
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.

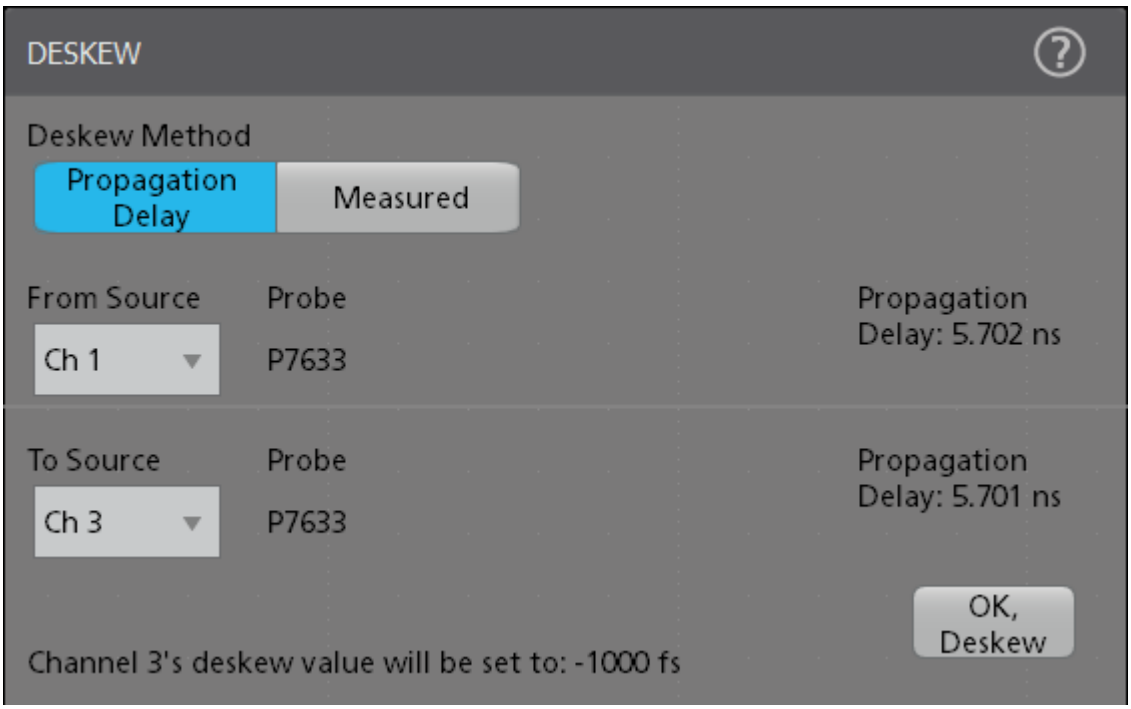
Quick visual method

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

1. Connect all probes that you want to deskew.
2. Connect any probes to the appropriate deskew fixture.
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.

Automatic Deskew window options

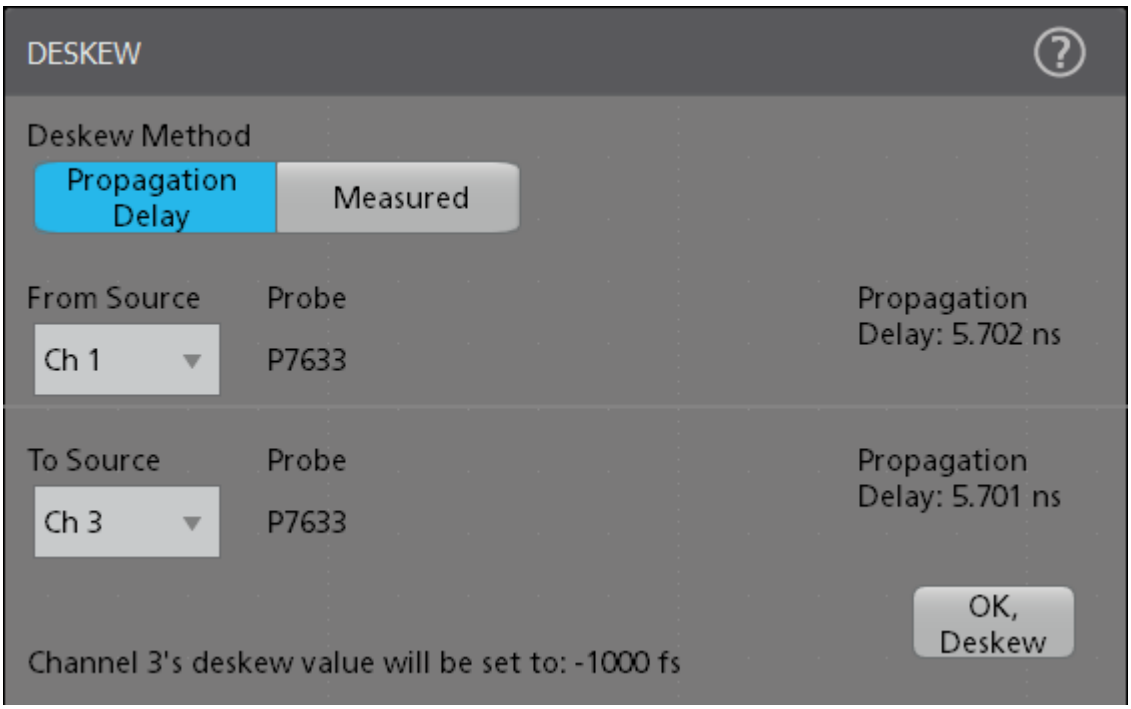
You can use the Deskew window menu to access propagation delay or a measured deskew algorithm to adjust channel timing alignment. Use the following procedure to set up your cables or probes before selecting an automatic deskew option.



1. Connect all probes or cables that you want to deskew.
2. Connect any probes to the appropriate deskew fixture.
3. Ensure you use the Differential Fast Edge connection on the instrument as the signal source for the deskew fixture.
4. Turn on (display on screen) the connected channels that you want to deskew.
5. Use the Vertical scale and Position controls to adjust scale and position for each channel so the fast edge signal takes up 8 divs on screen.
6. Adjust the horizontal scale so the fast edge signal takes up 8 divs on screen.
7. Adjust the vertical and horizontal scale position to see the fast edge signal takes up 8 divs on screen.
8. Double tap the channel badge to open **Vertical Settings**.
9. Click the **Other** panel.
10. Click the **Multi-Channel** button and the **Deskew** window will open.
11. Select **Propagation Delay** or **Measured** as the Deskew method.
12. Next, proceed to either the Propagation Delay or Measured procedure that follows.

Propagation Delay method

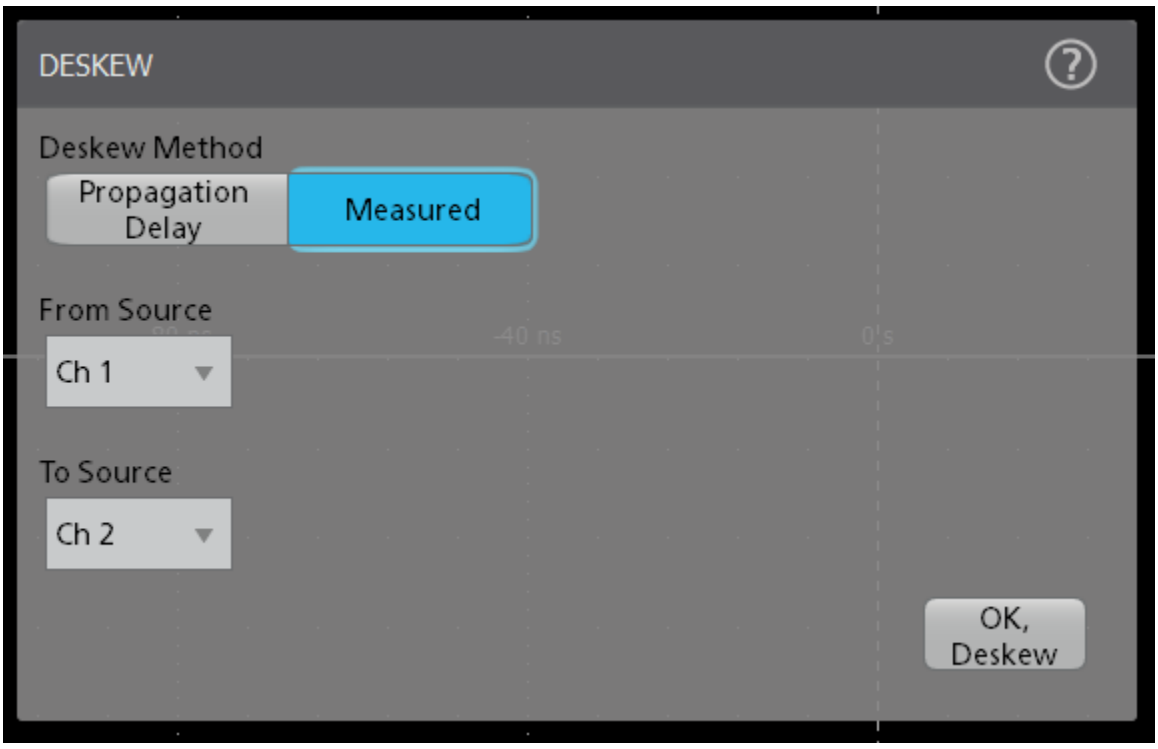
This is the default deskew method when a probe is connected. Deskew values will be adjusted based on the automatically detected propagation delay of the connected probe.



1. Select **Propagation Delay** as the Deskew Method.
2. Select **Measured.**
3. Set the To Source (Channel to be adjusted/deskewed).
4. Set the From Source (Reference channel).
5. Tap **OK, Deskew** and the system automatically runs the deskew algorithm.
6. If the deskew succeeds, a window opens showing the measured deskew value.
7. If the deskew fails, a window opens with an error message explaining why it failed. If this occurs, you will want to make adjustments and try again.

Measured method

This deskew method applies an algorithm to adjust channel timing alignment.



1. Determine the channel you want to use as your reference.
2. Double-tap the Channel badge of a channel other than the reference channel and tap the **Other** panel.
3. 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.
4. Repeat the previous steps for each additional channel you want to deskew.
5. Tap **OK, Deskew** and the system automatically runs the deskew algorithm.

Deskew topic. (on page 92)

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 Display Port or HDMI 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.

Calibrate the oscilloscope touchscreen when connected to external monitor

When the Windows oscilloscope is connected to an external monitor with extend screen mode enabled, and touchscreen of the oscilloscope is disabled, follow below procedure to calibrate the touchscreen.

1. Connect an external monitor to the oscilloscope. Refer Connect an external monitor or projector (on page 76).
2. From the Start menu, search for **Calibrate the screen for pen or touch input**, and select it.
3. In the Tablet PC Settings window, select the **Display** corresponding to the oscilloscope.
4. Click **Calibrate** to start the touch calibration process to explicitly identify the touch-enabled oscilloscope screen.
5. Follow the on-screen instructions to complete the calibration.
6. Save the calibration data when prompted.

Once calibration is completed, Windows correctly recognizes the touch screen.

Analog channel operating basics

Acquiring a signal

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

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

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

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

3. Tap the channel button to add the channel waveform to the waveform view and add a channel badge to the settings bar. A channel button lights when its channel is on.
4. Use the Vertical to scale and position the waveform vertically on the screen. The should be highlighted with the active channel's color. Dragging the waveform handle also positions the waveform.
5. Use the Vertical Settings menu to change the offset. Tap **Offset** and then use a multipurpose control to adjust the offset.
6. Use the Horizontal control to scale and position the waveform horizontally on the screen and to set the record length. Dragging the reference icon also positions the waveform.
7. Use the Horizontal menu to set the record length and sample rate (when in Manual mode).
8. 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.

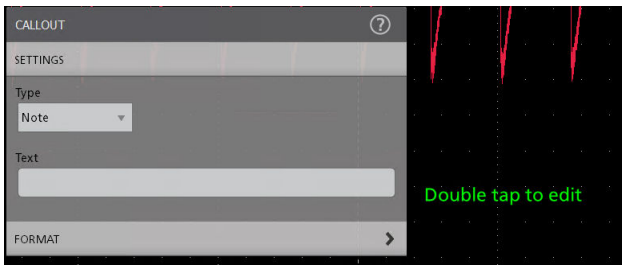
Add a callout to a view

Tap the **Add New... Callout** button to add callouts at the default position, but you can relocate it later.

1. Tap the **Add New... Callout** button on the upper right of the results bar. The oscilloscope adds a text placeholder with default text as *Double tap to edit* at the center of the screen. However, you can move it around to place it at your choice.
2. The default callout field wraps text at about 15 characters. Use a mouse to select the callout and select any point from the available 8 touch points to resize the callout field to a larger size, to autowrap the callout text for the longer callout.



3. Double-tap or Double-click on the callout text top to open the callout Settings configuration menu.



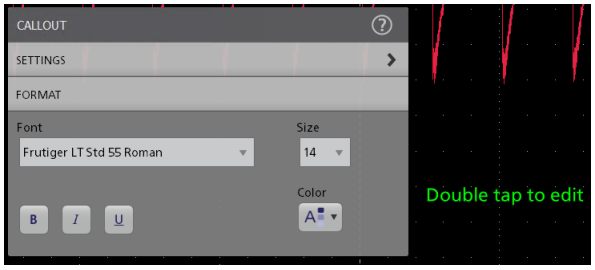
4. Select the type of callout from the drop-down list. The types of callout available are as follow:

Callout Type	Description
Note	Adds a callout as a text.
Arrow	Adds a callout text with an arrow.
Rectangle	Adds a callout text with an arrow pointing to a rectangle. The rectangle can be dragged to any position.
Bookmark	Adds a callout text with a bookmark added for either Waveform view (ch, math, ref), Time Trends. Specify the source channel from the drop-down list and X-position of the callout.

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

6. If you used the on-screen keyboard to enter text, tap the **Enter** button on the keyboard to close the keyboard and show the callout on the screen.
7. Use the format menu controls to set font type, size, color and other characteristics. See Text Settings configuration (on page 528) for details.



8. Tap outside the callout Settings configuration menu to close it.
9. To move a callout, touch, and drag the callout text to a new position.

Delete a Callout

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

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

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. 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.

Add a measurement

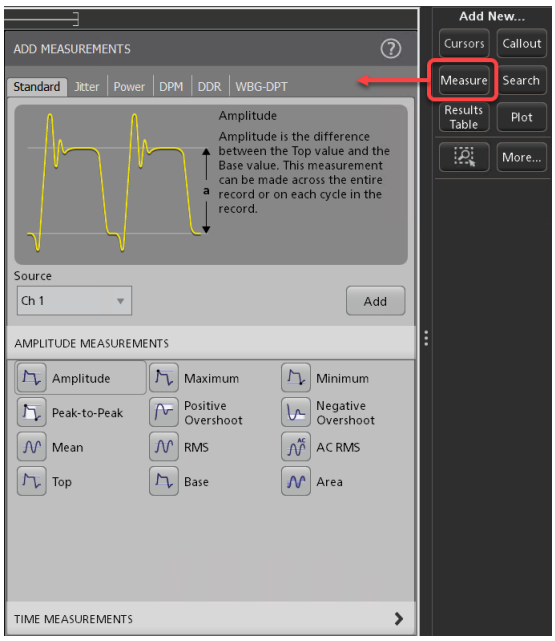
Use this procedure to select and add measurements.

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



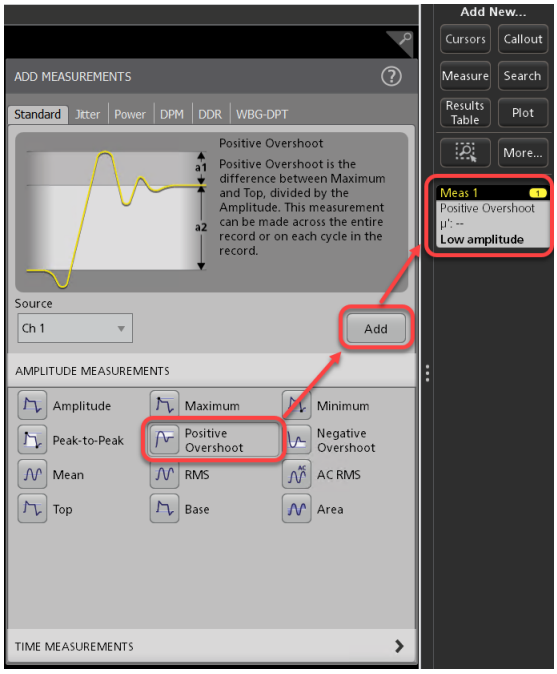
NOTE: Waveforms do not need to be displayed to be used for measurements, as long as the channel or waveform badge is on the **Settings** bar and is acquiring the signal to measure.

2. Tap the **Add New...Measure** button to open the **Add Measurements** configuration menu or drag the Measure button onto a waveform in the waveform display area to automatically set the source.

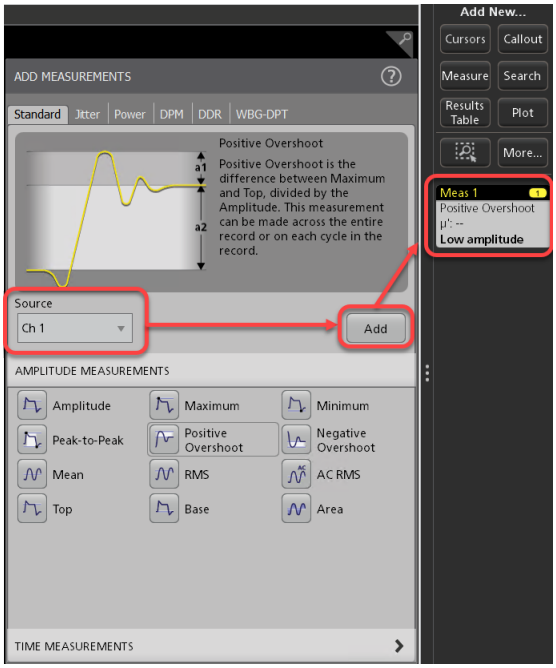


NOTE: If the menu shows tabs other than **Standard**, then optional measurement types have been installed on the instrument. Select a tab to show the measurements for that option.

3. Tap the **Source** field and select the measurement source.
4. Select a measurement category panel, such as **Amplitude Measurements** or **Time Measurements**, to display measurements for those categories.
5. Some optional measurement packages are available in this menu as well like DJA.
6. 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.



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



9. Tap outside the **Add Measurements** menu to close the menu.
10. 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 91).
11. Tap the Help icon on the menu title for more information on settings.

Add a math, reference, or bus waveform

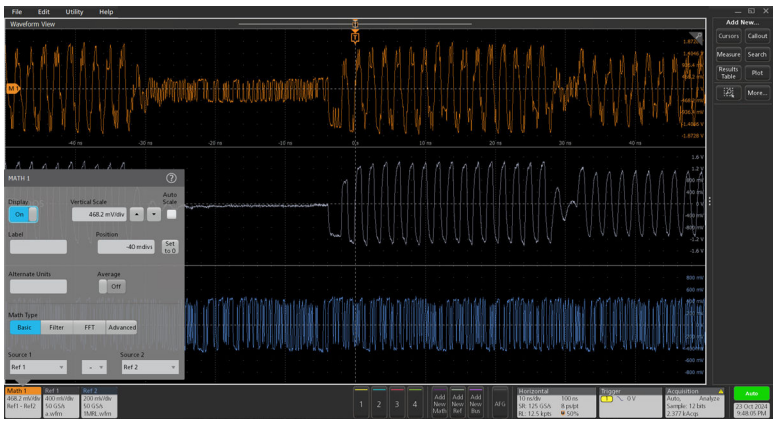
Math waveforms let you create new waveforms based on operations between two or more waveforms or by applying equations to waveform data. A reference waveform is a static waveform record displayed for comparison. Bus waveforms let you view and analyze serial or parallel data.

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.

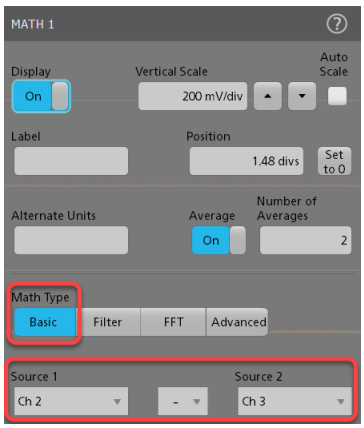


2. The instrument adds the waveform to the Waveform view, adds a Waveform badge to the Settings bar, and opens a configuration menu. This example shows adding a Math waveform.



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.

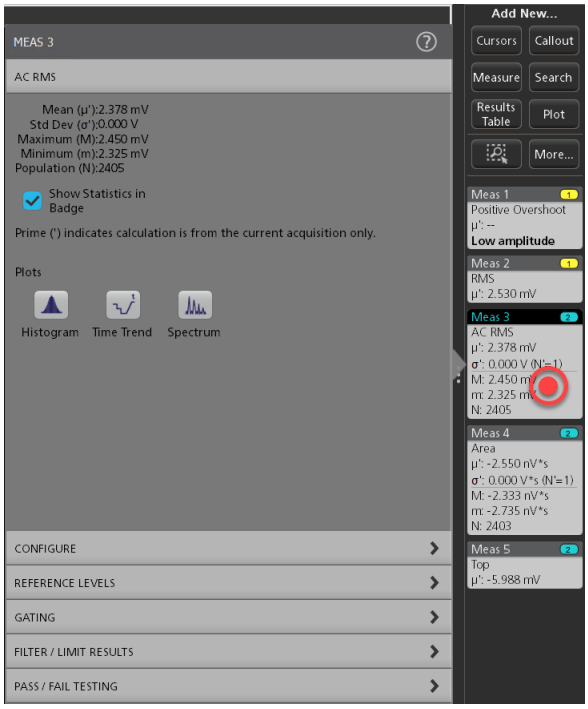


4. When adding a Reference waveform, the instrument displays a **Recall** configuration menu. Navigate to and select the reference waveform file (*.wfm) to recall, then tap the **Recall** button. The instrument displays the Reference waveform.
5. Double-tap a math, reference, or bus badge to check or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

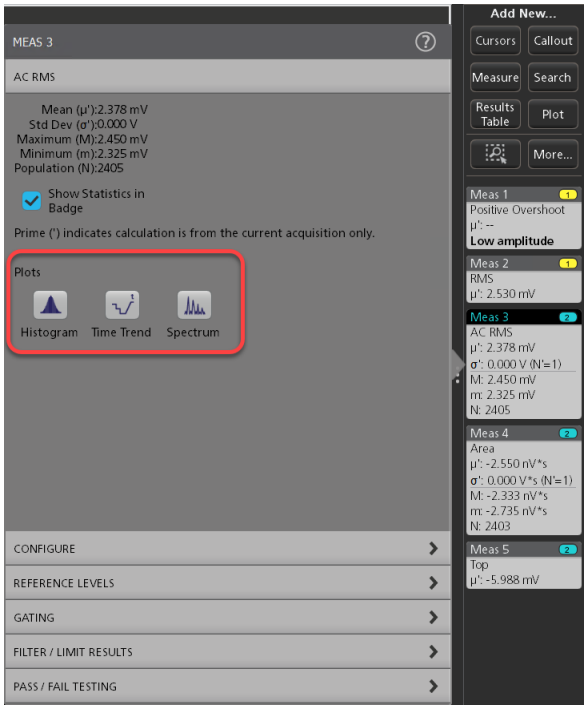
Add a plot of a measurement

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, and other supported plots. Available plots depend on the measurement.

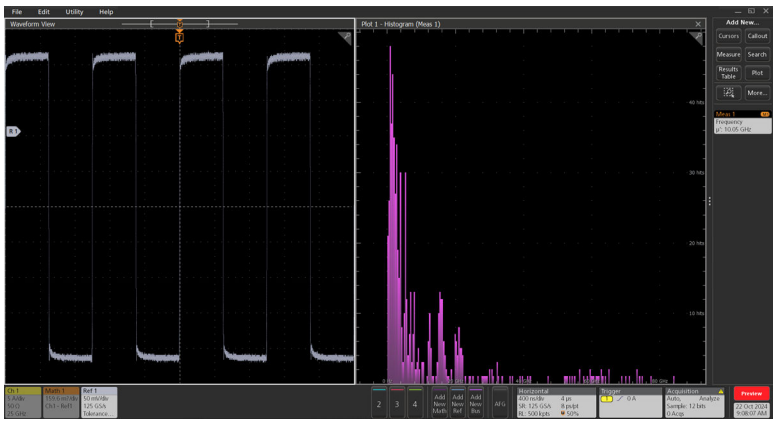
1. Double-tap a Measurement badge to open the **Meas** configuration menu.

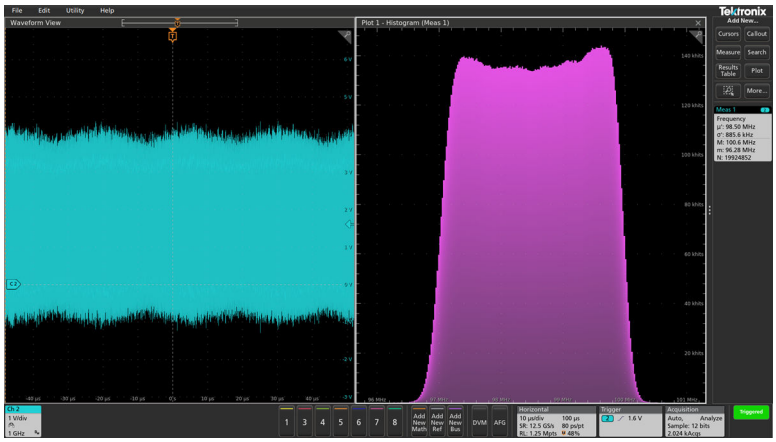


2. Tap a **Plots** button to add that plot for the measurement to the screen.



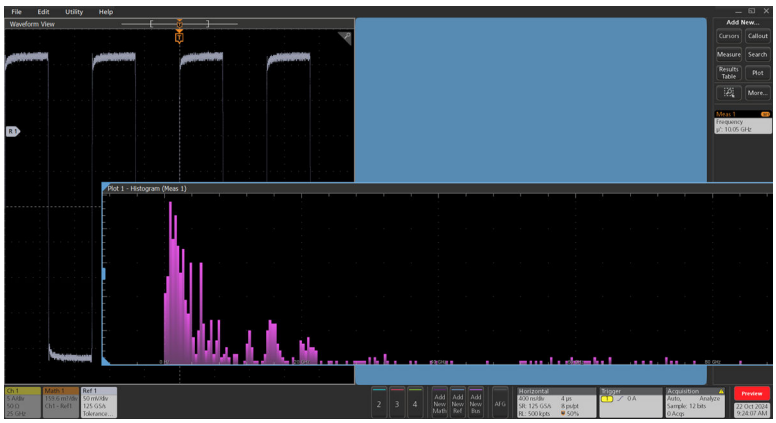
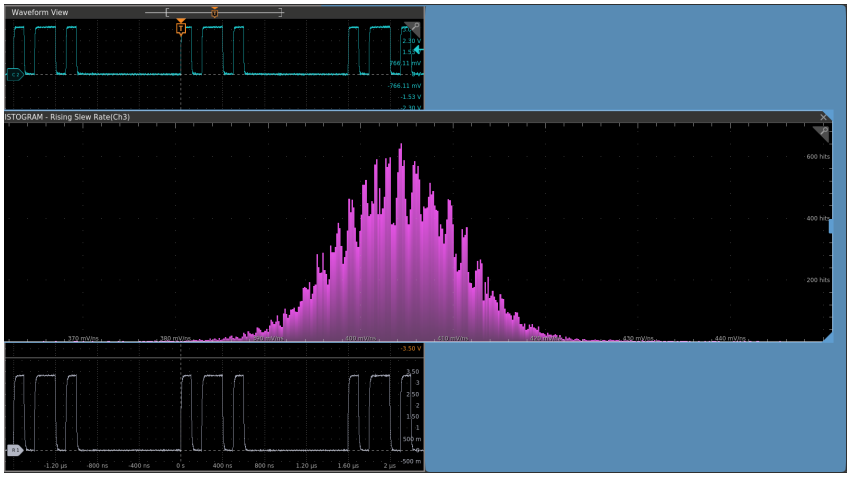
The following shows adding a Histogram plot.





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.

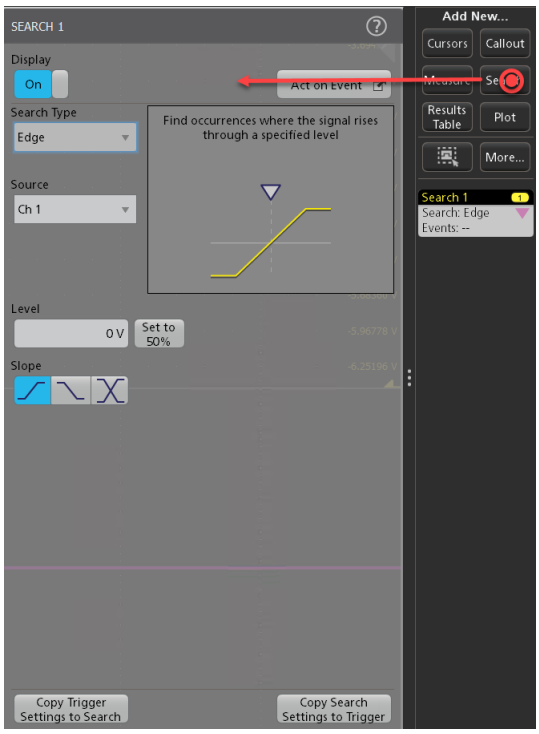
Add a Search

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

You can search on analog 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.

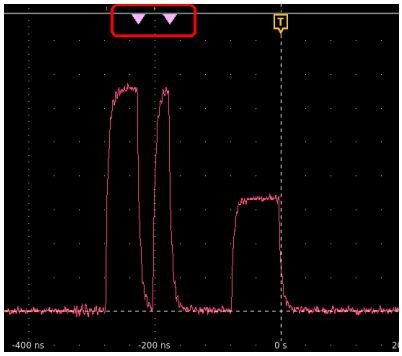
1. Tap the **Add New...Search** button to open the Search configuration menu.



2. 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).

3. 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.



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

6. 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.
7. 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.

Add mask testing to an Eye Diagram or BER Eye Contour plot

Use this procedure to add mask test segments to an eye diagram plot. To perform mask testing on eye diagrams, either draw a mask or recall a mask file. To draw a mask, refer Create a Mask (on page 114).

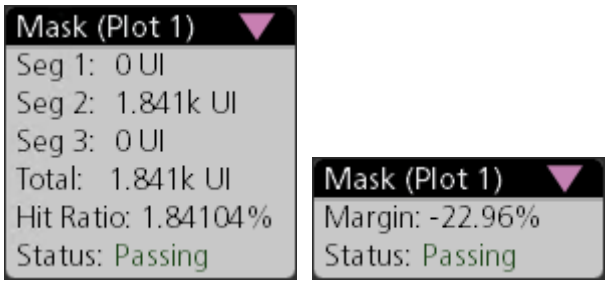
NOTE: Mask Testing can also be done on BER Eye Contour plot by following similar steps as given here.

Prerequisites: To recall a mask configuration file (.msk), the file must match the time scale (unit interval) and vertical settings of the active eye diagram.

1. Select the eye diagram plot on which the mask will be applied.
2. Tap **File > Recall** from the File menu.
3. From the **Recall** menu, select Mask as the file type.
4. Select the mask configuration file to recall, and then tap **Recall**. Alternatively, drag and drop the mask file onto the eye diagram plot. See Eye Diagram plot configuration menu (on page 352).

   When a mask configuration file is recalled, a Mask badge is added to the Results bar with **Mask Test** set to **Off**.

   Tap the **Mask Test** badge to open the Mask Test configuration menu. Selecting **Mask Margin**, displays the mask margin value in the badge. Selecting **Mask Hits**, displays the segment mask hits in the badge.



   The **Mask** badge displays the number of unit interval hits per segment and the pass/fail status. You can expand the Mask badge when acquisition is stopped to show the Navigation buttons when **Mask Test** is **Mask Hits**.

   When **Autofit Mask to Minimize Hits** box is checked, the mask violations are displayed in sample hits.

   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.

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 Test** badge in the Results bar to open the **Mask Hits** menu and set the **Fail if Greater Than** value. See Mask Test badge configuration menu (on page 290).

Analog view of Bus

Use this procedure to see the digital waveforms in analog view.

1. Tap the **Add New Bus** button in the Settings bar.

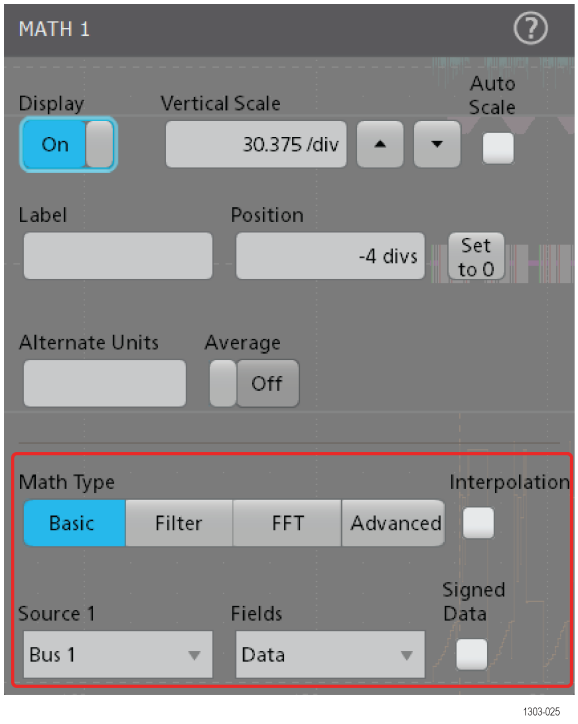


   The instrument adds the Bus waveforms to the Waveform view, adds a Bus badge to the Settings bar, and opens a configuration menu.

2. Tap the **Add New Math** button in the Settings bar.

   The instrument adds the Math waveforms to the Waveform view, adds a Waveform badge to the Settings bar, and opens a configuration menu.

3. Set the **Math Type** as **Basic** from Math configuration menu.
4. Select the **Bus <x>** you want to decode from **Source 1** drop-down list. Source 1 sets the available fields in the menu.
5. Select the **Field** that you want to plot for the selected bus.



6. Select the other fields and panels to refine the waveform parameters.
7. Tap the Help icon for more information on math waveform settings.
8. Tap outside the menu to close the menu.

Autoset to quickly display a waveform

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

1. Connect the probe with the signal of interest to an available channel.
2. Double-tap the **Trigger** badge and set the trigger source to that of the signal of interest.
3. Connect any other associated signal(s) to available channel input(s).
4. Add the channel waveforms to the Waveform view. See Add a channel waveform to the display (on page 80).
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 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. Access the Autoset panel in **Utility > User Preferences > Autoset.** See User Preferences (Utility menu) (on page 322).

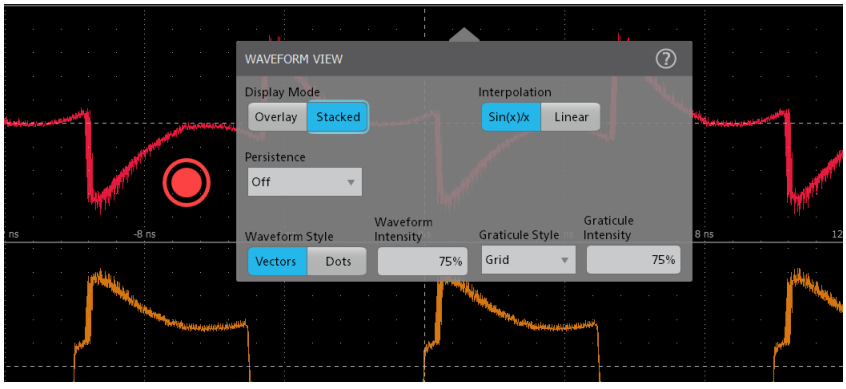
Autoset guidelines

- 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.

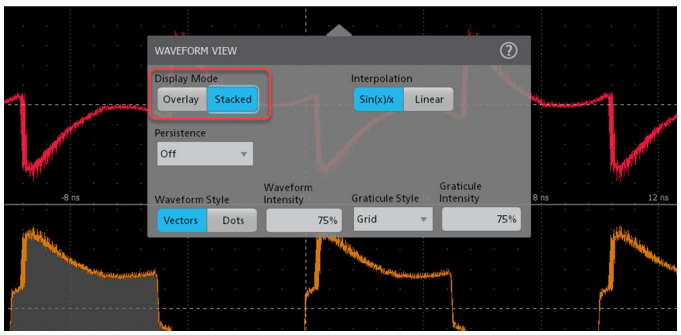
Change waveform view settings

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.

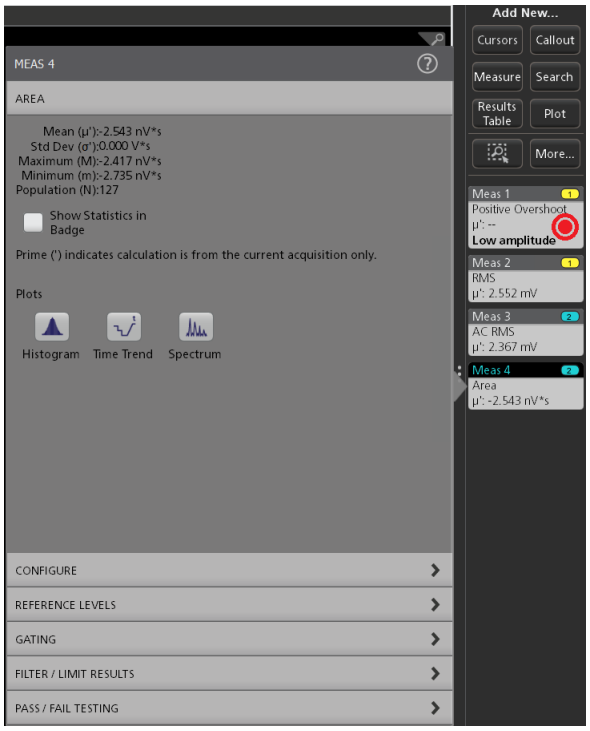


3. Use the other controls to set the waveform interpolation algorithm, waveform point persistence, style, and intensity, and graticule style and intensity.
4. Tap the **Help** icon on the menu title to open the Waveform View menu help topic for more information on the waveform view parameters.
5. Tap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

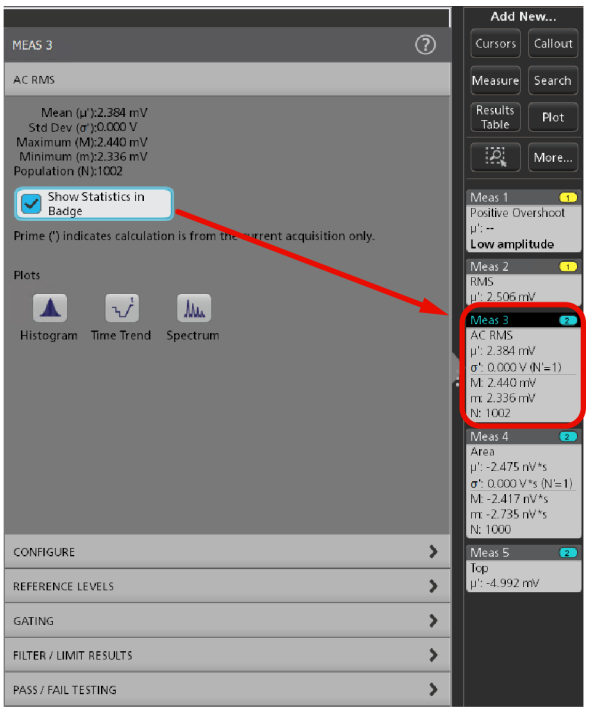
Configure a measurement

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).

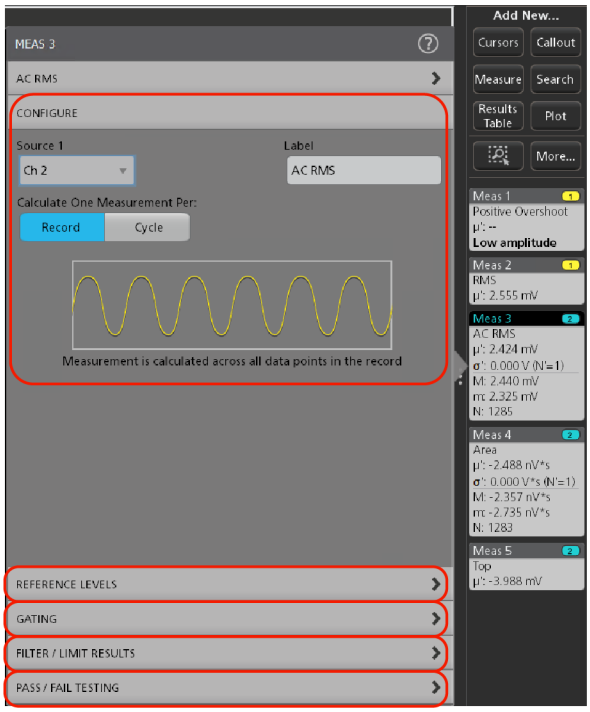
1. Double-tap a measurement badge to open its **Measurement** configuration menu.



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.

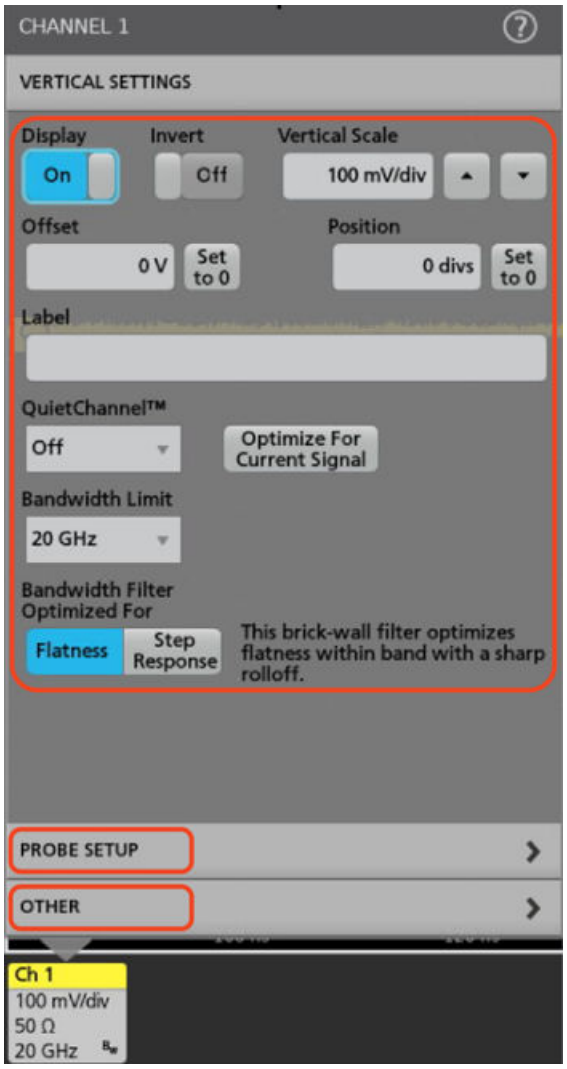
Configure channel and waveform settings

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.

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

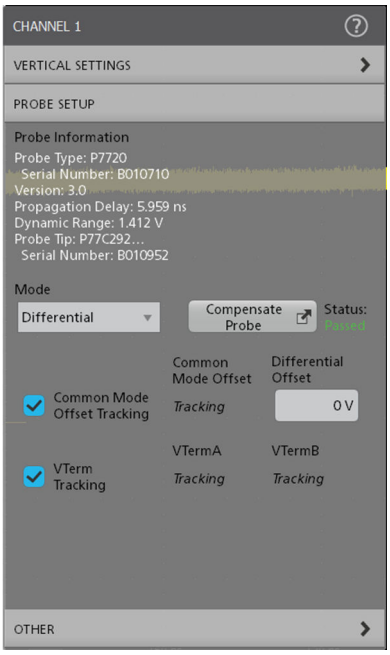
1. Double-tap a **Channel** or **Waveform** badge to open the configuration menu to the Vertical Settings panel. Displayed fields and controls can change depending on menu selections.



Field or control	Description
Display	Enables **(On)** or disables **(Off)** displaying the channel waveform in the Waveform View window.
Invert	Reverses the amplitude polarity of each sample point. When On, Invert adds a down arrow symbol to the channel badge. See Invert channel guidelines (on page 200).
Vertical Scale	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Offset	Sets the channel signal vertical offset. When there is an offset, the channel handle displays it in the waveform view.
Set to 0	Sets the channel signal vertical offset to 0 units.

Field or control	Description
Position	Double-tap the field to set the vertical position using the virtual keypad.
Set to 0	Tap to set the waveform zero volt level to the center of the slice or waveform view.
Label	Double-tap the field to add a label to the channel display using the virtual keypad.
Quiet Channel	Select the Quiet Channel setting from the drop-down list as **QC1** - **QC7** or **Off**. Default is off. See QuietChannel technology guidelines (on page 198).
Optimize for Current Signal	Tap to execute an algorithm that optimizes the Quiet Channel setting based on signal currently being driven into the channel.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Bandwidth Filter Optimized For	Tap to select a bandwidth filter that is optimized for flatness or step response. **Flatness** 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. **Step Response** selects a Bessel-Thompson filter that minimizes overshoot with a gradual rolloff.

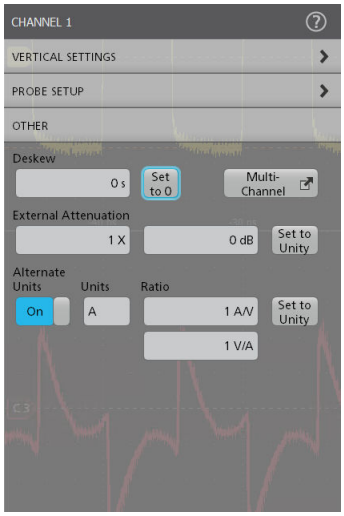
2. To set up your probe, tap the **PROBE SETUP** panel. The fields and controls in the Probe Setup panel vary with the type of probe connected to the channel. A P77 probe is shown in this example, for other probes, see the probe user documentation.



Field or control	Description
Probe Information	View probe information such as probe type, serial number, and its attenuation.

Field or control	Description
Probe Compensation Status	View the probe compensation status: Default, Pass, or Fail.
Compensate Probe	Tap to display the probe compensation dialog. This is only available for probes that support automatic compensation.
Mode	Select the probe operating mode from the drop down list (Differential, Common Mode, A, or B).
Offset	Enter the A or B offset value. This value is the same as the Offset in the Settings panel.
Differential Offset Tracking	When checked, the Differential Offset Tracking mode is Tracking. When unchecked, the Differential Offset Tracking is Manual.
Differential Offset	When in manual mode, enter the Differential Offset value.
Common Mode Offset	When in manual mode, enter the Common Mode Offset value.
Common Mode Offset Tracking	When checked, the Common Mode Offset Tracking mode is Tracking. When unchecked, the Common Mode Offset Tracking is Manual.
VTerm Tracking	When checked, the VTerm mode is Tracking. When unchecked, the VTerm is Manual.
VTermA	When in manual mode, enter the VTermA value.
VTermB	When in manual mode, enter the VTermB value.

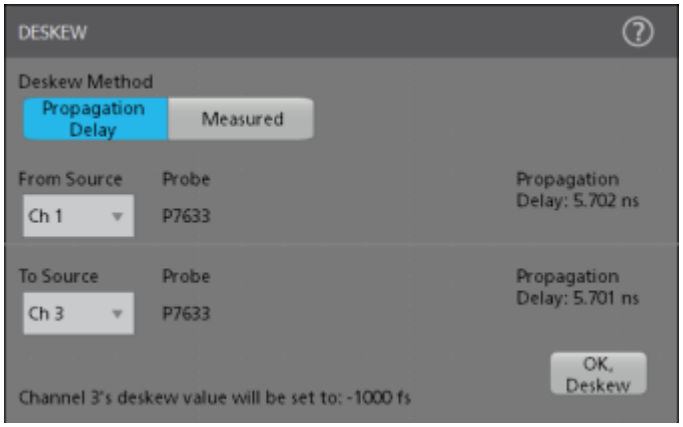
3. To set up deskew, external attenuation, or alternate units, tap the **OTHER** setup panel.



Field or control	Description
Deskew	Allows setting the channel deskew value.
Set to 0	Sets the channel deskew to 0.

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

4. When **Multi-Channel** in the **Other** panel is selected, the DESKEW configuration menu is displayed. You can select Propagation Delay or Measured as the deskew method.



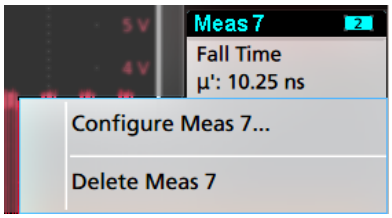
Field or control	Description
From Source	Selects the From Source to Deskew.
Probe	Displays the probe name or a drop-down list to select the probe connected to the From Source.
Propagation Delay	Displays the propagation delay of the probe shown in the Probe control.
To Source	Selects the To Source to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

5. Tap outside the menu to close the menu.

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.



NOTE: You can undo a measurement delete.

3. The second way to delete a Measurement or Search badge is by flicking it off the right edge of the display. Flicking to the left from the right edge of the display recovers the badge.

NOTE: Badge recovery is possible within 10 seconds only of removal.

Display a Acq Trend plot

Use this procedure to display an acq trend plot.

To display an acq trend plot you must be taking a measurement.

1. Double-tap a measurement badge. The Measurement configuration menu is displayed.
2. Tap the **Acq Trend** plot button.

NOTE: Plot rendering resets when there is change in the oscilloscope settings such as the vertical settings, horizontal, acquisition and trigger etc.

When oscilloscope acquisition is stopped and restarted again, the plot will continue to render without resetting.

Acq trend plot does not support Save/Recall session feature.

Cursors can be enabled on the plot.

The Acq Trend 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 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.

Note: The eye diagram plot displays Unit Interval (UI) and Unit Amplitude (UA) reference dots at the normalized ±0.5 UI and ±0.5 UA positions, indicating the 1 UI horizontal (time) and 1 UA vertical (amplitude) reference axes. These markers provide visual reference for eye opening, mask alignment, and measurement interpretation.

Display a Transfer Function plot

Use this procedure to display a transfer function plot of two measurements.

Prerequisite: Two active measurements with valid spectral data for which you want to plot a transfer function.

1. Tap **Add New... Plot** to open the Plot configuration menu.
2. Tap **Transfer Function**.
3. Tap the **Numerator** drop-down and select the measurement to use as the output (Y(f)) from the list.
4. Tap the **Denominator** drop-down and select the measurement to use as the input (X(f)) from the list.
5. Tap **Add**. The instrument displays the transfer function plot in a separate plot window.
6. Double-tap in the Plot view to open a configuration menu for that plot.

Display an FFT math waveform

Use this procedure to display an FFT math waveform.

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

1. Tap **Add New Math** to create a math waveform and open the Math configuration menu.
2. Tap **Source** and select the signal source from the list.
3. Set Math Type to **FFT**. The FFT of the waveform is displayed in an FFT Math waveform view.
4. Double-tap on the FFT waveform display to open a configuration menu to further refine the FFT display. See Math FFT plot configuration menu (Math waveform) (on page 362).

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 an XY or XYZ plot

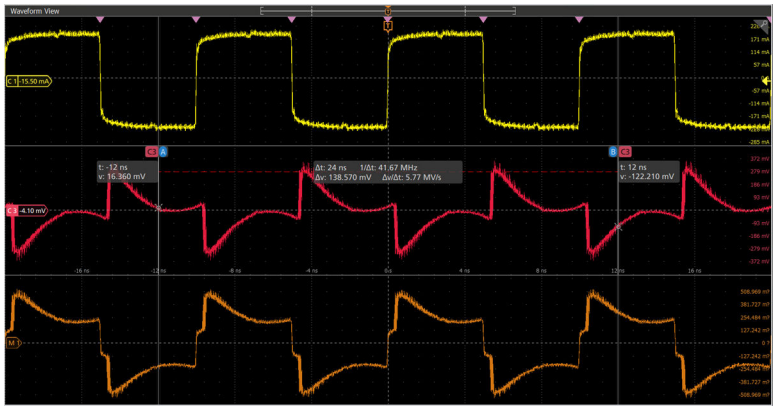
Use this procedure to display an XY or XYZ plot.

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.

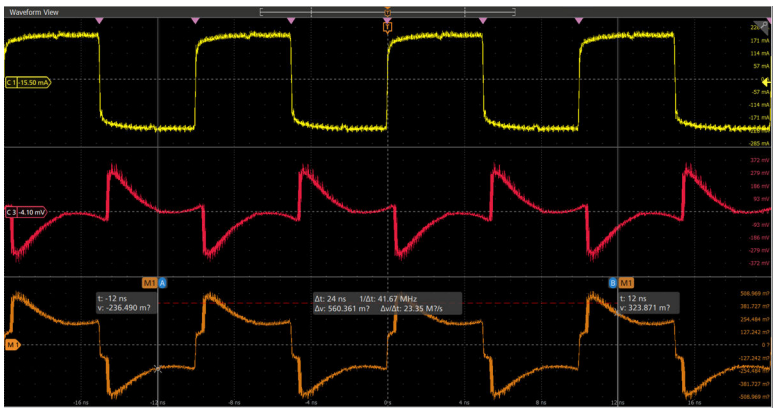
Display and configure cursors

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. Polar cursor readouts are available through the cursor configuration menu for XY and XYZ plots.

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 button. The cursors are added to the display.

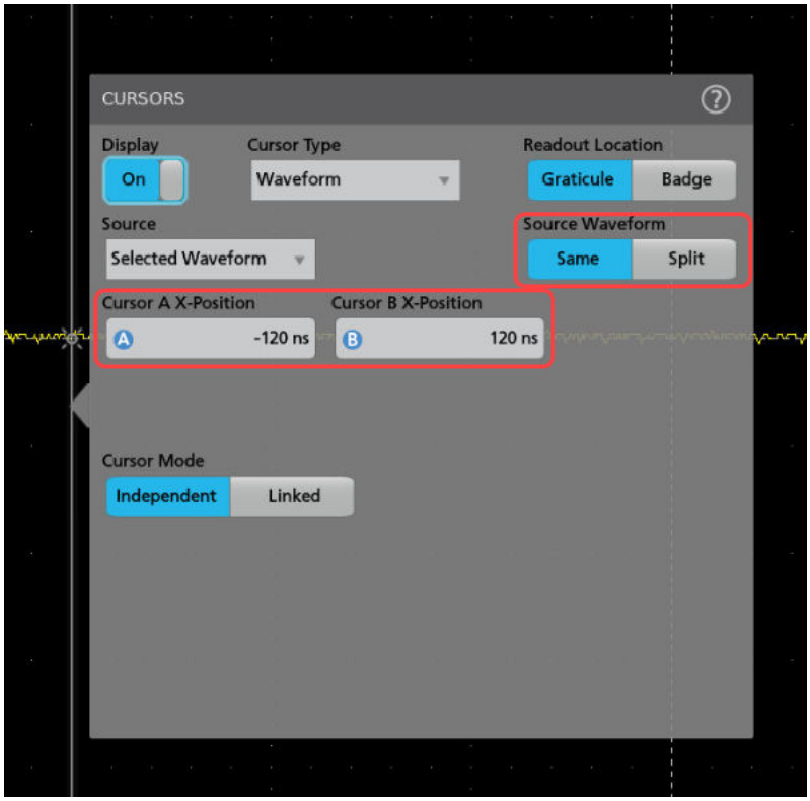


3. Use Multipurpose Knobs **A** and **B** to move the cursors, or touch and drag a cursor. Cursors show readouts that show position and difference measurements between the cursors.
4. To move the cursors to a different channel or waveform, just tap in that waveform graticule.

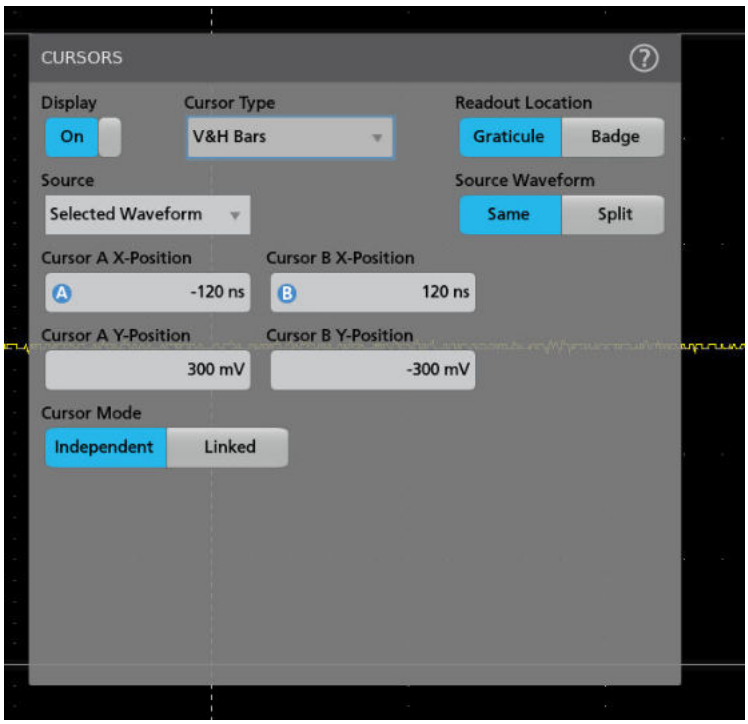


5. To further configure cursors, double-tap on either cursor line or the cursor readouts to open the **Cursors** configuration menu. For example, tap the Cursor type to select the cursors to display, such as Waveform, V Bars, H Bars, and V&H Bars. See Cursor configuration menu (on page 254).

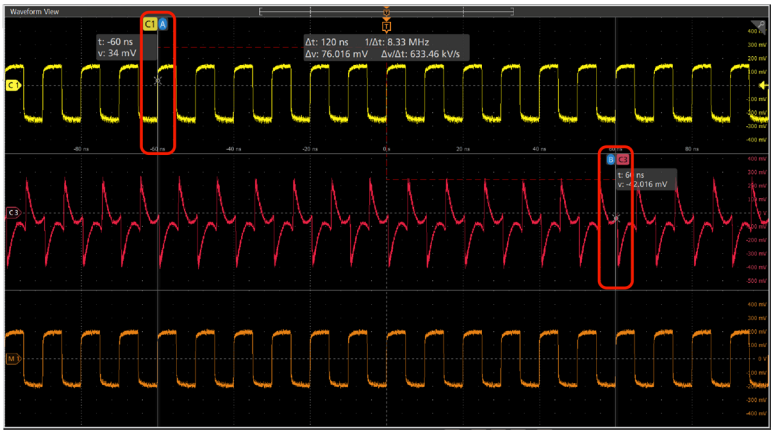
   The cursor configuration menu in the waveform view.



   The cursor configuration menu in an XY plot.



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



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

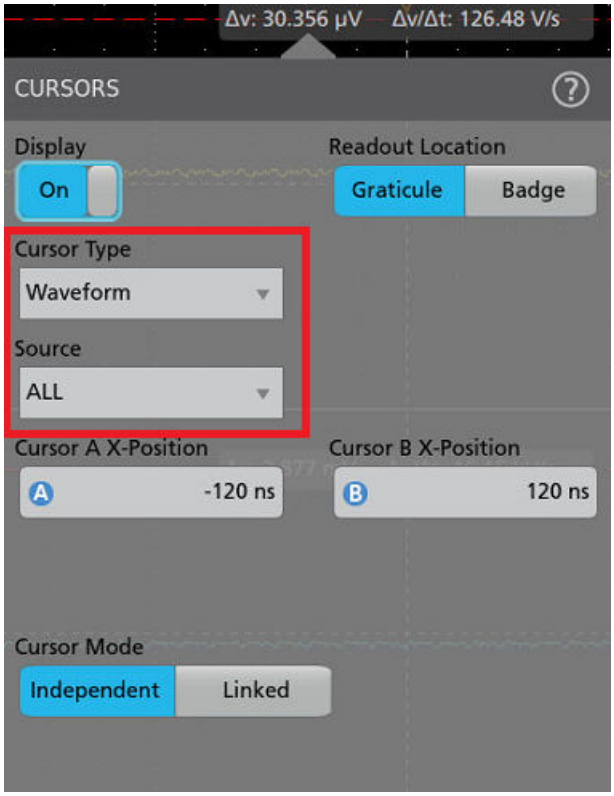
Display and configure Multi-source cursor readouts

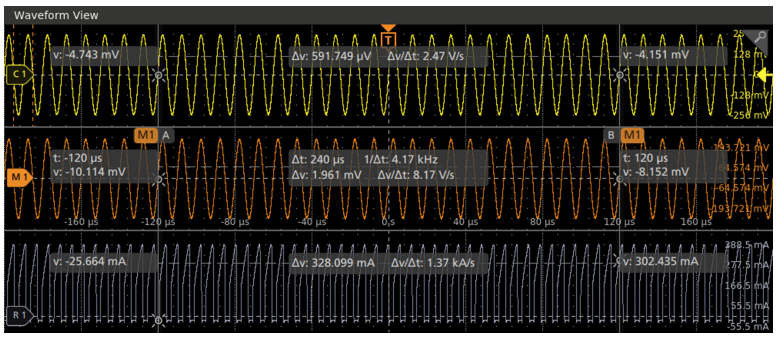
Multi-source cursor readouts are displayed for waveform cursor type. They display amplitude and delta values between cursors for all the active sources in Stacked display mode. The readouts also display the time and delta values on the selected source. The cursor time values are common for all the active sources.

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

   Use Multipurpose Knobs **A** and **B** to move the cursors, or touch and drag a cursor. Cursors display position and difference measurements readouts between the cursors.
3. Double-tap on the cursor line or the cursor readouts to open the **Cursors** configuration menu.
4. Tap **Cursor Type** and select **Waveform** from the drop-down menu.
5. Tap **Source** and select **ALL** from the drop-down menu.

   All includes time domain analog waveforms like math, reference, trend, etc.





6. Tap the Help icon for more information on the menu settings.
7. To stop displaying cursors,

a. push the front panel **Cursor** button,

b. press and hold to open the right-click menu and turn cursors off, or

c. open the Cursors configuration menu and set **Display** to **Off**.

Multi-source cursor readouts guidelines

The waveform in each slice displays the crosshairs where the cursors intersect the waveform.

If the cursor **Readout Location** is set to **Graticule** in the configuration menu:

- The cursor readouts display the amplitude and amplitude delta values for all the active sources.
- The cursor readouts for the selected waveform shows both time and amplitude values and deltas.
- If you select a different waveform or channel, the cursor readouts and horizontal in-graticule labels moves to the selected waveform.

If the cursor **Readout Location** is set to **Badge** in the configuration menu:

- All active waveforms in the Waveform View have cursor readout results associated with them, even if they are overlaid in a slice.

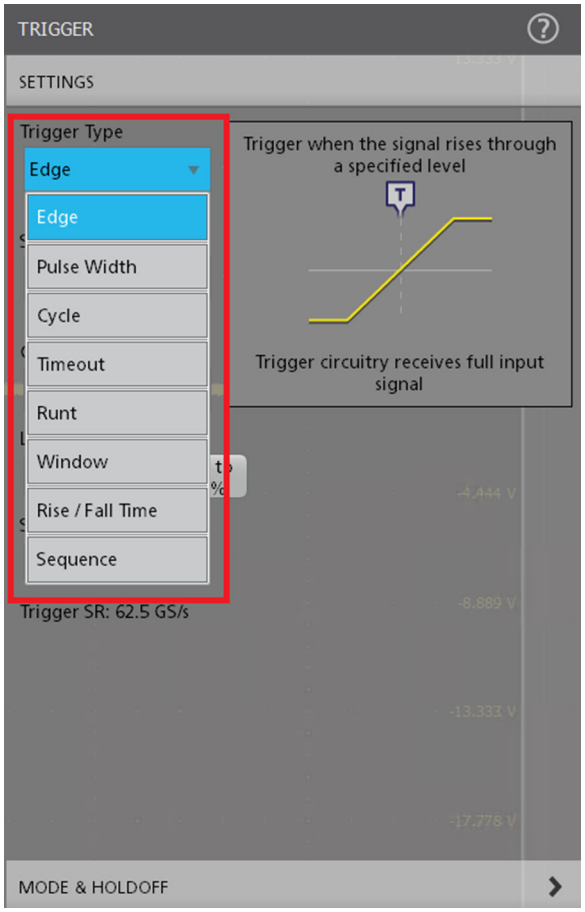


- The first section of the badge shows the delta time information.
- The second section of the badge shows the time and amplitude values associated with Cursor A.
- The third section of the badge shows the time and amplitude values associated with Cursor B.
- The fourth section of the badge shows the delta amplitude information.

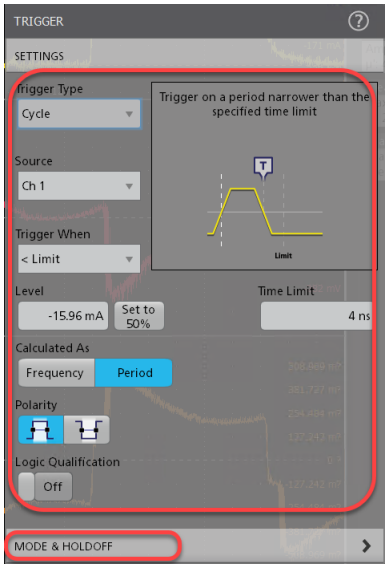
How to trigger on a signal

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.



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



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

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.

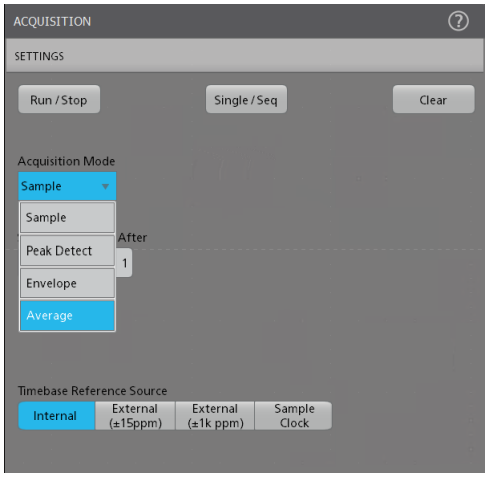


2. To stop an acquisition, tap **Run/Stop** again or push the **Run/Stop** button.
3. To take a single acquisition, double-tap the Acquisition badge and tap **Single/Seq** in the Acquisition configuration menu, or push the **Single/Seq** button on the front panel.
4. The color of the **Run/Stop** and **Single/Seq** buttons on the front panel indicate the acquisition status (green = acquiring, red = stopped).
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.

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.

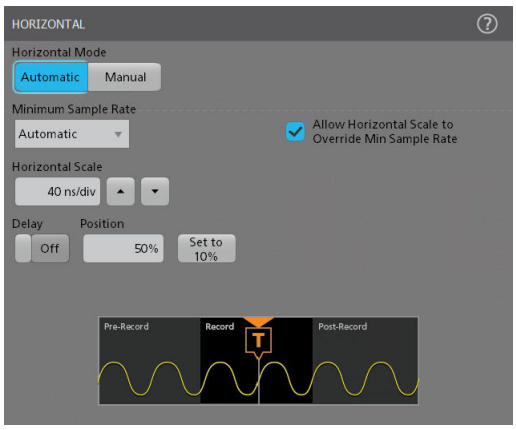


3. Tap the Help icon on the menu title for more information on these settings.
4. Tap outside the menu to close the menu.

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.

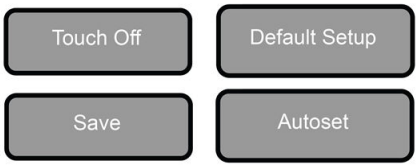


2. Use the menu selections to set horizontal parameters.
3. Tap the Help icon on the menu title for more information on these settings.

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).

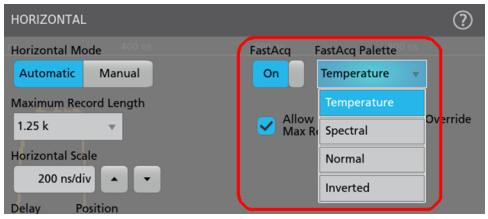


2. You can also select **File** > **Default Setup** to restore default settings.

Using FastAcq

Fast acquisition mode (FastAcq™) reduces the dead time between waveform acquisitions, enabling the capture and display of transient events such as glitches or runt pulses. FastAcq mode can also display waveform occurrences at intensity levels that reflect their rate of occurrence. The Horizontal badge displays **Analyze** when FastAcq is set to **Off** and **FastAcq** when it is set to **On**.

1. To use FastAcq mode, double-tap the **Horizontal** badge and tap **FastAcq** to toggle FastAcq to On.
2. To display waveform with an intensity that reflects the rate-of-occurrence, double-tap the Horizontal badge, tap the **FastAcq Palette** field, and select a display palette from the drop-down.



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



If FastAcq mode is on and you attempt to activate a feature that conflicts with this mode, FastAcq will be disabled. When the conflicting feature is turned off, FastAcq will resume in most cases. Enabling FastAcq mode disables Fast Frame mode and enabling Fast Frame mode disables FastAcq mode.

Setting waveform display parameters

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

Set 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.
   a. **Off** disables display persistence.
   b. **Auto** lets the oscilloscope automatically determine a persistence time for you.
   c. **Infinite** persistence continuously accumulates record points until you change one of the acquisition display settings. Use infinite persistence for displaying unique signal anomalies, such as glitches.
   d. **Variable** persistence accumulates record points for a specified time interval. Each record point decays independently according to the time interval. Use variable persistence for displaying infrequently appearing signal anomalies, such as glitches.

If you select Variable persistence, tap **Variable Persistence Time** and set the time using the multipurpose knob, or double-tap the field and use the virtual keypad to enter the time value.

3. Tap the Waveform Style buttons to set waveforms to draw as vectors (continuous lines) or dots.
   - **Vectors** displays the waveform with the waveform sample values connected using the selected interpolation method.
   - **Dots** displays the individual waveform sample values with no interpolation.
4. Tap the **Waveform Intensity** field and use the multipurpose knob to set the brightness of all waveforms.

Set the graticule style and intensity

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

1. Double-tap on an open graticule area to open the Waveform View configuration menu.
2. Tap the **Graticule Style** field to select a graticule style from the list.
   - **Grid** provides a grid, cross hairs, and frame on the instrument display.
   - **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 callout text on the screen.

Zoom in on waveforms

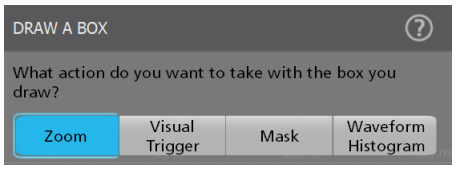
Use the zoom tools to magnify waveforms to view signal details.

Turn on Zoom mode

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

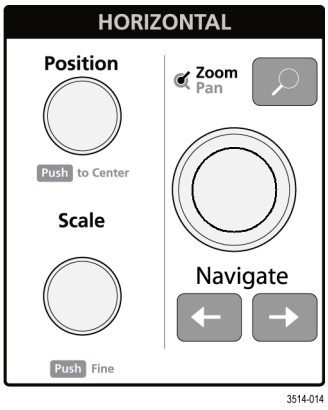
To enable Zoom mode, use one of the following methods:

1. Double-tap the **More** button and select the **Zoom** button on the **DRAW A BOX** menu window. This sets the Draw-a-Box function to Zoom mode.

   Tap the Draw-A-Box **Zoom icon** button at the up right of the **Results Bar**, then touch and drag in the Waveform view to draw a box on the area of interest. The 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 zoom icon touch and draw a box around an area of interest in the Waveform or Plot view to immediately display the zoomed waveform and the Zoom Overview window.
b. You can draw boxes in the Waveform view, most plots, and the Zoom Overview area.
c. The **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:
   - Zoom user interface (on page 58)
   - Using Wave Inspector front-panel controls for zoom (on page 112)

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 Zoom user interface (on page 58).

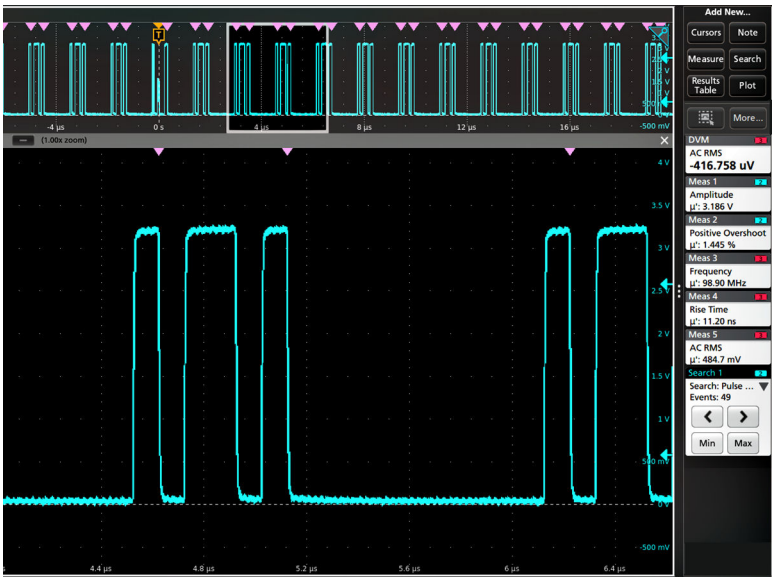
Zoom mode and Searches

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.



Search tables and Zoom mode

- When you select a row in the Search Results Table, Zoom mode adjusts the position of the zoom window to place the selected search event in the middle of the zoom window.
- If zoom was not already on, it is turned on with its previous zoom factor and the zoom window position adjusted to show the selected search event.
- When a Search Results Table is selected, Multipurpose knob **A** is assigned to the index column. Use knob **A** to scroll through the table and display the selected event in the zoom window.
- If you pan (move horizontally) the zoom window by dragging it, or by using the front panel **Pan** control, the search table highlights which (if any) search result is at the center of the zoom window.

For information on creating a Search, see Add a Search (on page 86).

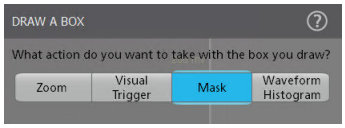
Mask testing

Mask testing determines how well your signals conform to expected signal quality and provides instant automated statistical analysis. A mask defines a portion of the instrument display that a signal must not enter. Mask testing determines how well your signals conform to expected signal quality and provides instant automated statistical analysis. A mask defines a portion of the instrument display that a signal must not enter. Mask testing is supported for analog channels, analog math waveforms, analog reference waveforms, Eye Diagram and BER Eye Contour plot.

Create a Mask on waveforms

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

1. Double-tap the **More** button and select the **Mask** button on the **DRAW A BOX** menu window. This sets the DRAW A BOX function to Mask mode.

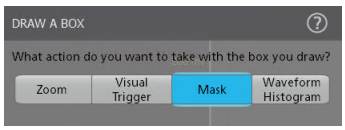


2. Tap outside the menu to close the **DRAW A BOX** menu. The Draw a Box icon is highlighted to show that it is in Mask draw mode.
3. Tap the **DRAW A BOX Mask** icon at the up right of the **Results Bar** to draw a rectangle on the waveform screen for the first segment to define. The segment created is associated with the channel to which it was added (in Stacked mode) or the active channel (in Overlay mode).
4. Continue drawing rectangles to add other mask segments:
   a. If in Stacked mode, you can draw mask segments in different waveform slices.
   b. If in Overlay mode, select a channel before creating segments for that channel.
5. When you are done drawing the mask segments, tap anywhere to end the segment draw function.
6. To draw segments again, tap the **DRAW A BOX Mask** icon again.
7. To change the shape of a segment, see and Edit visual trigger areas using the Area menu (on page 546).

Create a Mask on Eye diagram plot

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

1. Double-tap the **More** button and select the **Mask** button on the **DRAW A BOX** menu window. This sets the DRAW A BOX function to Mask mode.

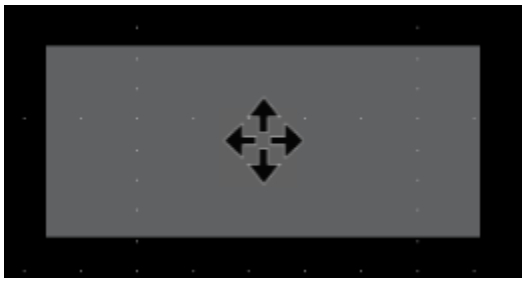


2. Tap outside the menu to close the **DRAW A BOX** menu. The Draw a Box icon is highlighted to show that it is in Mask draw mode.
3. Tap the **DRAW A BOX Mask** icon at the up right of the **Results Bar** to draw a rectangle on the eye diagram plot for the center segment..
4. When you are done drawing the mask segment, tap anywhere to end the segment draw function. You can draw only one center segment on an eye diagram plot.
5. To delete a mask segment:
   a. Touch and hold or right click on the segment.
   b. Select **Delete Segment** from the menu.

Edit mask segments on the screen

Use the touchscreen or mouse to edit a segment. You can move, resize, duplicate, or rotate a segment, add, delete, or adjust individual vertices. For eye diagrams, only center mask segment can be edited.

1. To move a mask segment, touch and drag the segment to a new position.



NOTE: You cannot drag a mask segment from one waveform slice to another while in Stacked display mode.

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

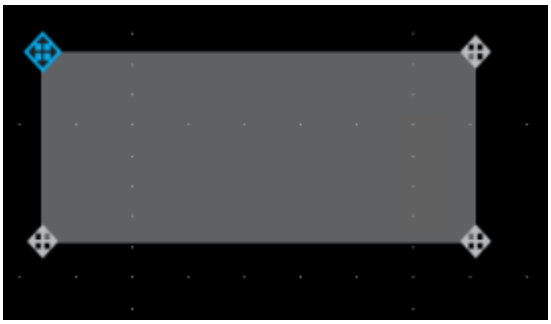


   b. Touch and drag a top, bottom, left, or right resize point to resize the segment vertically or horizontally.
   c. Touch and drag a corner resize point to resize both the length and height of the segment in the dragged direction.

NOTE: You cannot move a segment while in resize mode.

d. To exit resize mode, tap outside the mask segment. The instrument returns to normal operation.

4. To add vertices to or delete vertices from a mask segment:
   a. Tap a segment twice 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.



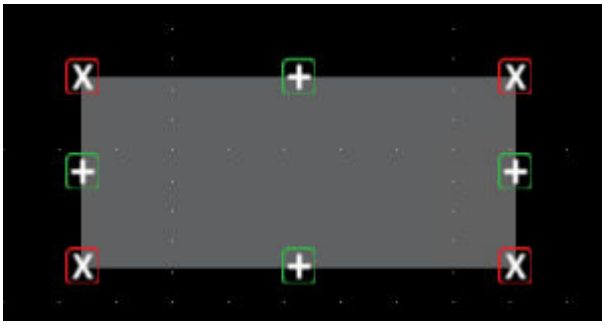
NOTE: A triangular segment does not draw the existing vertices with crosshairs, as they cannot be deleted; only the midway plus symbols are shown to let you add vertices to a triangle.

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

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

   d. To exit add/remove vertex mode, tap outside the mask segment. The instrument returns to normal operation.

5. To move vertices in a mask segment:
   a. Tap the segment three times to enable move vertex mode. The move vertex mode draws crosshairs on all vertices of the segment.



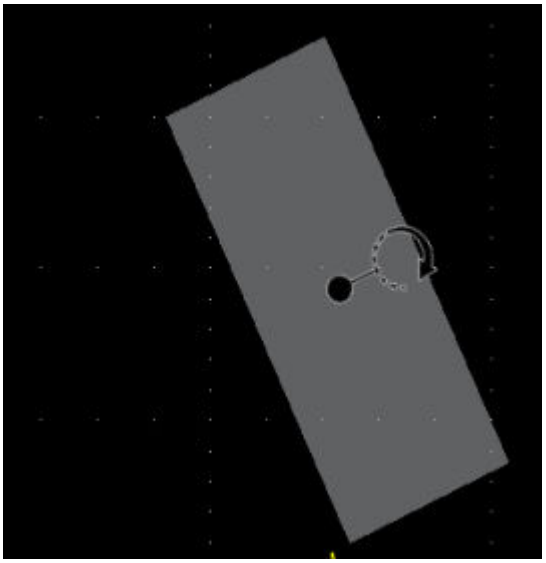
   b. Touch and drag a vertex to move it to a new position.
   c. To exit move vertex mode, tap outside the mask segment. The instrument returns to normal operation.

6. To rotate a mask segment:
   a. Tap four times in the segment to enable rotate segment mode.



b. Touch and drag the dot in the segment to rotate it.

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



c. To exit rotate segment mode, tap outside the mask segment. The instrument returns to normal operation.

NOTE: For an overshoot segment on an eye diagram, only the bottom point can be dragged vertically, and for an undershoot segment, only the top point can be dragged vertically.

Edit waveform mask segments using the Mask Definition menu

The Mask Definition menu allows you to edit the mask. You can add, move, resize, duplicate, delete, or adjust individual vertices, and rotating the segment. You have the choice to edit a mask by segments or waveform tolerance with the **Mask Defined By** options in the Mask Settings panel.

See Mask Definition Segment configuration menu (on page 287)

See Edit mask segments on the screen (on page 115) for information on graphically editing a segment on the screen.

CAUTION: Do not toggle between the **Mask Defined By** options without saving your mask settings. When **Waveform Tolerances** is selected, a mask generated by the **Segment** settings are replaced with a mask generated by the tolerance fields.

See for more information on the mask settings.

1. To change the shape of a segment using the Mask Settings menu:

a. Double-tap on the segment to edit.
b. Select **Segments** under the **Mask Defined By** options.
c. Tap the **Shape** field.
d. Select the shape to which to change the segment. The segment changes shape immediately.

2. To change the height of a segment using the Mask Settings menu:
   a. Double-tap on the segment to edit.
   b. Select **Segments** under the **Mask Defined By** options.
   c. 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 segment.
3. To change the vertical center of a segment using the Mask Settings menu:
   a. Double-tap on the segment to edit.
   b. Select **Segments** under the **Mask Defined By** options.
   c. 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 segment.
4. To change the width of a segment using the Mask Settings menu:
   a. Double-tap on the segment to edit.
   b. Select **Segments** under the **Mask Defined By** options.
   c. 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 segment.
5. To change the horizontal center of a segment using the Mask Settings menu:
   a. Double-tap on the segment to edit.
   b. Select **Segments** under the **Mask Defined By** options.
   c. 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.
6. To flip a segment vertically using the Mask Settings menu:
   a. Double-tap on the segment to edit.
   b. Select **Segments** under the **Mask Defined By** options.
   c. Tap the **Flip Vertical** button. The segment immediately flips vertically around its center position.
7. To flip a segment horizontally using the Mask Settings menu:
   a. Double-tap on the segment to edit.
   b. Select **Segments** under the **Mask Defined By** options.
   c. Tap the **Flip Horizontal** button. The segment immediately flips horizontally around its center position.
8. To rotate a segment using the Mask Settings menu:
   a. Double-tap on the segment to edit.
   b. Select **Segments** under the **Mask Defined By** options.
   c. Tap the **Rotate** field and use the **A** knob to change the value. The segment 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 segment, where zero (0) degrees is the position when the segment was created. You can rotate a segment from 0° to 360°.
9. To add a point (vertex) to a segment using the Edit Vertices menu:

a. Double-tap on the segment to edit.
b. Select **Segments** under the **Mask Defined By** options.
c. Tap the **Edit Vertices** panel.
d. 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 segment.
e. Tap **Insert Point**. A new vertex is added to the segment, marked with an X, and is highlighted in the list.
f. Continue adding vertices as needed.

10. To delete a point (vertex) from a segment using the Edit Vertices menu:
    a. Double-tap on the segment to edit.
    b. Select **Segments** under the **Mask Defined By** options.
    c. Tap the **Edit Vertices** panel.
    d. Select the vertex in the **Define Area** list that you want to delete. Selecting a vertex highlights that vertex on the segment.
    e. Tap **Delete Point**. A new vertex is added to the segment, marked with an X, and is highlighted in the list.
    f. Continue deleting vertices as needed.
11. To move individual vertices in a segment using the Edit Vertices menu:
    a. Double-tap on the segment to edit.
    b. Select **Segments** under the **Mask Defined By** options.
    c. Tap the **Edit Vertices** panel.
    d. Select the vertex in the **Define Area** list that you want to move. Selecting a vertex highlights that vertex on the segment.
    e. 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.
12. To reset the shape using the Edit Vertices menu:
    a. Double-tap on the segment to edit.
    b. Select **Segments** under the **Mask Defined By** options.
    c. Tap the **Edit Vertices** panel.
    d. Tap **Reset Points**. The shape changes to a triangle centered in the display or slice.
13. To save the segment edits:
    a. Tap the **Save** panel.
    b. Tap **Save Mask**. Only edits to the segment settings are saved.
    c. Tap outside the menu to exit.
14. To change the vertical tolerance of a segment using the Mask Settings menu:
    a. Double-tap on the segment to edit.
    b. Select **Waveform Tolerances** under the **Mask Defined By** options.
    c. Tap the **Vertical Tolerance** field and enter the tolerance value as units of the current channel settings (volts, amps, and so on). Or double-tap on the field and use the **A** knob to change the value.
    d. Tap the **Update Mask Now** button to immediately change the vertical tolerance of the segment.
15. To change the horizontal tolerance of a segment using the Mask Settings menu:
    a. Double-tap on the segment to edit.
    b. Select **Waveform Tolerances** under the **Mask Defined By** options.
    c. Tap the **Horizontal Tolerance** field and enter the tolerance value as units of the current channel settings (volts, amps, and so on). Or double-tap on the field and use the **A** knob to change the value.
    d. Tap the **Update Mask Now** button to immediately change the horizontal tolerance of the segment.
16. To save the waveform tolerance edits:

a. Tap the **Save** panel.
b. Tap **Save Mask**. Only edits to the waveform tolerance settings are saved.
c. Tap outside the menu to exit.

Edit eye mask segments using the Mask Definition menu

The Mask Definition menu allows you to edit the central mask segment. You can add, move, resize, duplicate, delete, or adjust individual vertices, or rotate a segment. You have the choice to also convert the coordinate type between absolute and relative. Overshoot and undershoot segments can also be added/removed or configured by setting the limits.

To edit the mask using the mask definition menu, double tap on the central segment.

1. To change the shape of a central segment using the Mask Settings menu:
   a. Tap the **Shape** field.
   b. Select the shape to which you want to change the segment. The segment changes shape immediately.
2. To change the height of a central segment using the Mask Settings menu:
   a. Tap the **Height** field.
   b. Enter the height value as units of the current coordinate type 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 segment.
3. To change the vertical center of a central segment using the Mask Settings menu:
   a. Tap the **Vertical Center** field.
   b. Enter the vertical center value as units of the current coordinate type 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 segment.
4. To change the width of a central segment using the Mask Settings menu:
   a. Tap the **Width** field.
   b. Enter the width value as units of the current coordinate type 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 segment.
5. To change the horizontal center of a central segment using the Mask Settings menu:
   a. Tap the **Horizontal Center** field.
   b. Enter the 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.
6. To flip a central segment vertically using the Mask Settings menu:
   a. Tap the **Flip Vertical** button. The central segment immediately flips vertically around its center position.
7. To flip a central segment horizontally using the Mask Settings menu:
   a. Tap the **Flip Horizontal** button. The central segment immediately flips horizontally around its center position.
8. To rotate the central segment using the Mask Settings menu:
   a. Tap the **Rotate** field and use the A knob to change the value. The central segment 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 segment, where zero (0) degrees is the position when the segment was created. You can rotate a segment from 0° to 360°.
9. To add a point (vertex) to the central segment using the Edit Vertices menu:

a. Tap the **Edit Vertices** panel.
b. Select the vertex in the Define Area list counter-clockwise from where you want to insert a new vertex. Selecting a vertex highlights that vertex on the central segment.
c. Tap **Insert Point**. A new vertex is added to the central segment, marked with an X, and is highlighted in the list.
d. Continue adding vertices as needed.

10. To delete a point (vertex) from a central segment using the Edit Vertices menu:
    a. Tap the **Edit Vertices** panel.
    b. Select the vertex in the Define Area list that you want to delete. Selecting a vertex highlights it on the segment.
    c. Tap **Delete Point**.
    d. Continue deleting vertices as needed.
11. To move an individual vertices in a segment using the **Edit Vertices** menu:
    a. Tap the **Edit Vertices** panel.
    b. Select the vertex in the Define Area list that you want to move. Selecting a vertex highlights it on the central segment.
    c. 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.
12. To reset the shape using the Edit Vertices menu:
    a. Tap the **Edit Vertices** panel.
    b. Tap **Reset Points**. The shape changes to a hexagon centered in the display or slice.
    c. To change the coordinate type using the Mask Settings menu, tap the **Absolute** or **Relative** option under **Coordinate Type**. All the values are updated according to the new units.
13. To add overshoot/undershoot segments, select the **Overshoot/Undershoot Segments** checkbox.
    - When checked, overshoot/undershoot segments are added with a width of 1UI.
    - When unchecked, overshoot/undershoot segments are removed.
14. To change the Bottom Y coordinate of the overshoot segment:
    a. Tap the **Upper limit** field.
    b. Enter the value as units of the current coordinate type or double-tap on the field and use the **A** knob to change the value. The bottom coordinate of the overshoot segment changes accordingly.
15. To change the Top Y coordinate of the Undershoot segment:
    a. Tap the **Lower limit** field.
    b. Enter the value as units of the current coordinate type or double-tap on the field and use the **A** knob to change the value. The top coordinate of the overshoot segment changes accordingly.

Signal Integrity Modeling (SIM)

The Signal Integrity Modeling (SIM) software option combines flexibility with a modern, intuitive user experience to set a new standard for on-scope signal integrity modeling and analysis. Define as many simulations as you like, shift reference planes, and explore complex signal paths directly from your oscilloscope.

The Signal Integrity Modeling feature is based on Tektronix's trusted SDLA software. SIM is integrated into the oscilloscope and accelerates setup with simplified workflows (Simple De-embed, Simple Embed), touch-optimized controls, and built-in automation.

SIM provides precise de-embedding and embedding of signal path components to enhance measurement accuracy, reveal true DUT signal behavior, and design systems that perform reliably in their intended environment. With it, you can move measurement reference planes using virtual test points, rapidly simulate signal path changes, and compensate for the frequency response of cables, probes and test fixturing, addressing the needs of a wide range of high-speed serial, RF, and power electronics tests.

SIM enables side-by-side comparisons of multiple circuit configurations, which delivers faster and deeper insights for complex modeling scenarios.

Many standards require that equalization is applied to the signal before measurements are taken. SIM Advanced (option SIMA) provides CTLE, FFE, and DFE equalization modeling tools to perform receiver equalization.

Key features

- Physical Model de-embed environment – multi-block
- Simulation Model embed environment – multi-block
- Virtual test points to view signals at impractical probe locations
- Simplified workflows for faster setup
- Time and frequency-domain plots for S-parameters and test points
- Touch-optimized controls – click and drag blocks and test points
- Save and export filter file results for running within Tektronix automated compliance software and third-party software applications
- Supports nearly unlimited number of simulations and test points simultaneously
- S-parameter transformations and comparisons for rapid validation
- Receiver equalization - FFE, DFE, and CTLE (requires SIMA)
- Transmitter equalization - add de-emphasis or pre-emphasis (requires SIMA)
- Automatic Tektronix oscilloscope and probe modeling

Requirements and installation

The SIM and SIMA options are available for the Tektronix 7 Series DPO. See the Signal Integrity Modeling (SIM) datasheet on tek.com for ordering information.

To install a SIM license, go to **Help > About > Install License**. Select the license file and tap **Open**. The license will get added to the **Installed license** list.

Model overview

SIM requires the definition of two models: a Physical model and a Simulation model. The measured waveform is transferred from the Physical model, which represents the de-embedded measurement path, to the Simulation model, which represents the embedded system, through a Thevenin-equivalent source. This ensures the simulation sees the waveform with the intended voltage and impedance characteristics. This dual-model approach separates real-world measurement effects from hypothetical circuit changes, making it easier to cleanly remove existing effects, simulate future scenarios, and confidently optimize system performance.

In both models, blocks are connected to each other via single-ended and differential connections. The blocks of the Physical Model and Simulation Model change depending on the selected workflow.

Blocks and test points can be added to both models as components and measurement elements, and then configured to form a circuit based on the desired requirements. Up to eight user-defined blocks can be added in both models. The input source can be set to single or dual input mode.

The gray blocks are required based on the workflow selected and cannot be deleted or edited from the models. You can add new blocks to represent various components of the signal path as it moves through the DUT. The blue blocks represent measurement elements used to measure the signal. All the blue blocks are part of the measurement system.

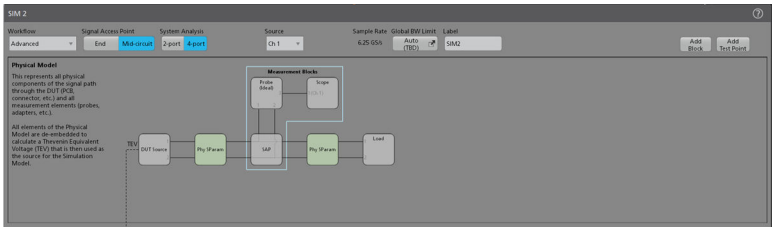
The test points in the model represent virtual probing locations that provide visibility into the link at multiple points within the circuit.

Once the models are defined and the user initiates Calculate SIM, SIM calculates a transfer function and generates an FIR filter for each test point. A corresponding math waveform is produced for each test point by applying the filter to the source waveform(s). The filters may also be exported as .flt files.

The SIM interface now supports improved workflow efficiency on Windows by operating as a standard window, allowing simultaneous interaction with both SIM and TekScope, including placement on a second monitor. Plot and Max Plot views are also available as independent windows for continuous waveform visibility during simulation. On Linux systems, the SIM interface remains tied to the main TekScope display and cannot be moved to a separate monitor.

The Physical Model

The Physical Model represents all the components and measurement blocks of the signal path through DUT, PCB, connectors, etc and all the measurement elements such as probes, oscilloscope, fixtures, and the portion of the channel between the Tx and the fixture. All the elements of the Physical Model are de-embedded to calculate Thevenin Equivalent Voltage (TEV) that is then used as source for the Simulation Model.



Physical Model example

The Physical Model can be configured for the Simple De-embed and Advanced workflow. All the blocks of the Physical Model are de-embedded to calculate the TEV.

The Simulation Model

The Simulation Model enables “what-if” exploration, showing how the waveform would look when embedded with an alternate circuit, without measurement effects, or both. The user interface and components of the simulation model changes depending on the selected workflow.

The Simulation Model can be configured for the Simple Embed and Advanced workflow with all the components and measurement blocks.

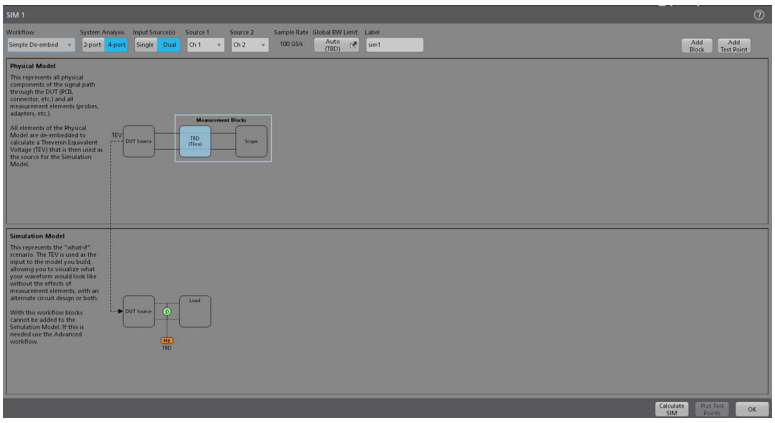


Simulation Model example

Simple De-embed workflow

In the Simple De-embed workflow, all the blocks of the physical model, including the DUT Source, Scope, and the Load or Measurement blocks are de-embedded to calculate the TEV. The de-embed block contains the measurement models that represent the actual hardware probe, fixtures, and other components used during waveform acquisition.

Select the **Workflow > Simple De-embed** from the SIM menu to run the Simple De-embed workflow.

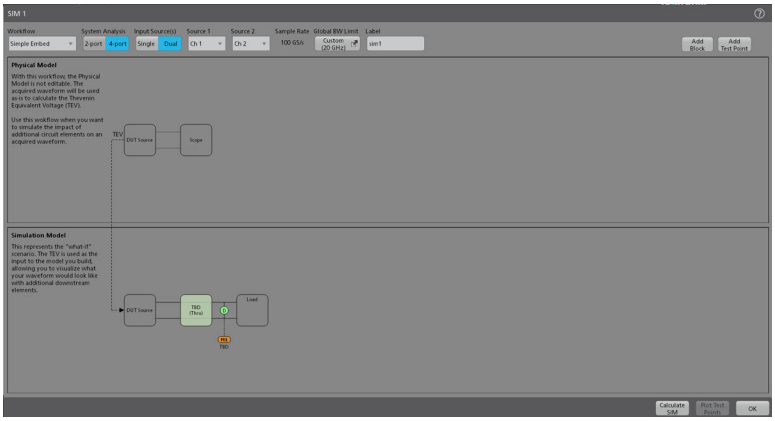


In the Simple De-embed workflow, DUT Source, Scope, and Load blocks are configured with a nominal 50 Ω impedance. If you want to configure these blocks with S-parameters then use the Advanced workflow.

Simple Embed workflow

For the Simple Embed workflow, the physical model is not configurable and the acquired waveform is used to calculate the TEV. In Simple Embed workflow DUT Source, Scope, and the load are configured with the nominal 50 Ω impedance. If you want to configure these blocks with S-parameters use Advanced workflow.

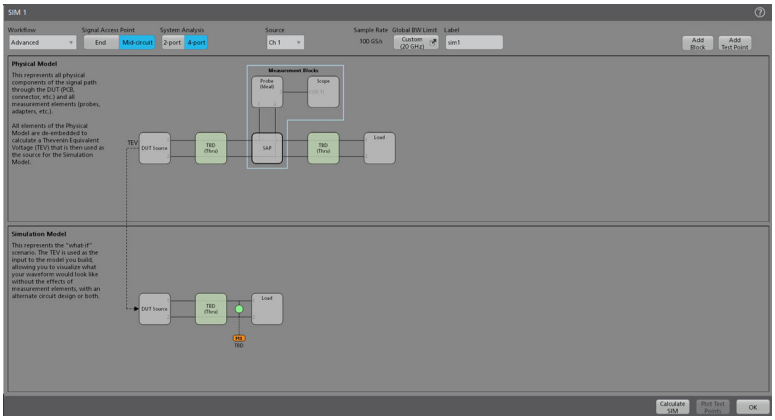
Select the **Workflow > Simple Embed** from the SIM menu to run the Simple Embed workflow.



Advanced workflow

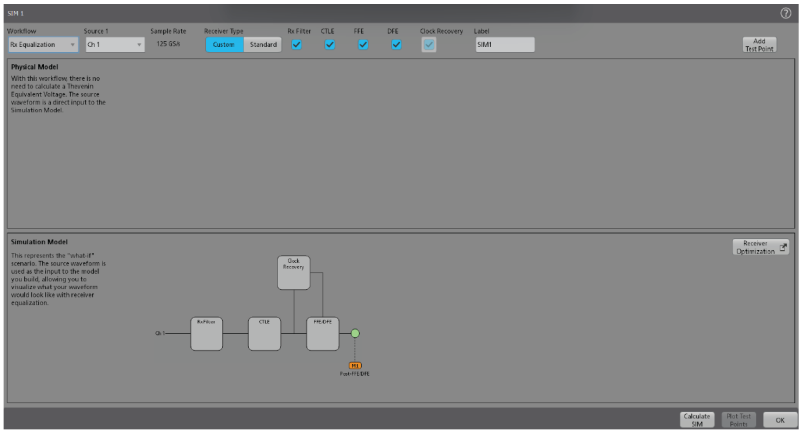
The Advanced workflow allows creation of a fully editable Physical Model and Simulation Model with either single-ended or differential sources. It also allows for mid-circuit probing through a Signal Access Point (SAP). With option SIMA installed, a receiver block can be placed to the right of the Load block for simulating receiver equalization, while the DUT Source block can be configured to perform transmitter equalization.

Select the **Workflow > Advanced workflow** from the SIM menu to run the Advanced workflow.



Rx Equalization workflow

The Rx Equalization workflow (requires option SIMA) simulates solely the receiver portion of a serial data link within the simulation circuit. It restores data stream integrity and recovers the embedded clock. The workflow can also serve as a reference receiver by operating at the minimal acceptable performance level defined by the applicable serial data standard. The Advanced workflow is required when also modeling the analog portion of the receiver—including the package and termination—as well as the channel and source using SIM blocks.



The Rx equalization compensates for the loss, crosstalk, reflection, or noise in the link. It attempts to optimize the signal-to-noise ratio of the eye opening, among other targets, in order to improve link performance in aspects such as bit error rate. In the oscilloscope measurement context, this equalization allows you to accurately simulate the signal timing and amplitude parameters at the receiver. Sometimes, this measurement point may be referred to as a virtual Rx, reflecting the simulated nature of the signals.

The receiver has a comparator or slicer that determines whether a symbol with value 0 or 1 is received in any unit interval. The exact timing at the slicer is determined by the clock recovery in the receiver. However, the virtual Rx is typically not directly accessible by probing or other methods, so the Rx equalization must be simulated in order to obtain adequate measurements.

Often, the signal at the input of the equalizer will have a closed eye. When correctly designed, the equalization will open the eye and increase the eye height, width, or both.

Create a SIM model

SIM helps you to create a physical or simulation model model step by step. Use below procedure to configure the blocks, models, and test points, and to plot and create and analyze the model.

1. Tap on **Add New SIM** to create a model.
2. Select the required **Workflow**. See SIM configuration menu (on page 483)
3. Select system analysis as **2-port** or **4-port**.
4. Select input sources to **Single** or **Dual**. See Configuring the blocks (on page 127)
5. Select the Source 1 and Source 2 from the drop down list. Source 2 is available when **System Analysis** is **4-port** and **Input Source(s)** is **Dual**. If **Source 1** is set to **None**, then by default **Source 2** is **None**.
6. Set the sample rate if **Source 1** is set to **None**.
7. Set the **Global BW Limit**. See Create a custom bandwidth limit filter (on page 132)
8. Set the **Signal Access Point** as **End** or **Mid-circuit** if **Workflow** is selected as **Advanced**. When signal access point set to end it enbles the input sources. See SAP configuration menu (on page 495).
9. Add and configure the blocks to the Physical/Simulation models.
   - See Block configuration menu (on page 486)
   - See Scope configuration menu (on page 498)
   - See Probe configuration menu (on page 494)
   - See Load configuration menu (on page 494)
   - See Clock Recovery configuration menu (on page 501)
   - See RX Filter configuration menu (on page 502)
   - See CTLE configuration menu (on page 502)
   - See FFE/DFE configuration menu (on page 502)
10. Add and configure the test point to the Physical/Simulation models. See Test Point configuration menu (on page 488).
11. Tap **Calculate SIM** to calculate the simulation model.
12. Tap **Plot Test Points** to plot the test points.

Duplicate a SIM Model

1. After creating a SIM model, locate the SIM badge at the bottom of the screen.
2. Right-click the SIM badge that you want to duplicate.
3. Tap **Duplicate**.
4. The instrument creates a duplicate SIM model with the same configuration settings as the original model.
5. A new SIM badge appears at the bottom of the screen.
6. Tap the new SIM badge to open and modify the duplicated SIM model as required.

Configuring the blocks

Single and Dual Input Source

You can use either Single or Dual inputs with SIM by selecting either Single Input or Dual Input mode. In Single Input mode, the differential signal may be viewed at each test point. Dual Input mode allows the viewing of individual inputs, differential, or common mode.

When to use each setting:

Single: Use when you have only one acquired signal to analyze, but still want the accuracy of a 4-port differential model. Example: You're testing one differential lane of a high-speed serial link, and channel S-parameters are provided as a 4-port file to preserve both differential and common-mode effects. The second input port in the model remains terminated but un-driven, so you still capture realistic crosstalk and mode-conversion behavior from the model.

Dual: Use when you have two acquired signals and want to drive both inputs in the 4-port model at the same time. Example: You're analyzing two active differential lanes in parallel (multi-lane link) or intentionally measuring crosstalk between them. Both lanes are fed from oscilloscope channels, reference waveforms, or math waveforms, allowing the model to simulate their combined effect.

Available input sources are:

- Channels: Live signals from active oscilloscope channels
- Math Waveforms: Active math results (Example: CH1–CH3 or waveforms calculated from other SIM test points)
- Reference Waveforms: Stored waveforms currently loaded and active
- None: Disables the input; useful when creating test point filter files on one instrument for use on another with a different supported sample rate

System Analysis

2-Port System Analysis: Model a single signal path such as a single-ended connection or a differential link already reduced to a 2-port S-parameter file for faster processing, simpler setup, and smaller files.

4-Port System Analysis: Model two signal paths (example: the ±legs of a differential pair) and preserve crosstalk alongside insertion and return loss.

Duplicate a Block

Use the Duplicate option to create a copy of an existing user-defined block.

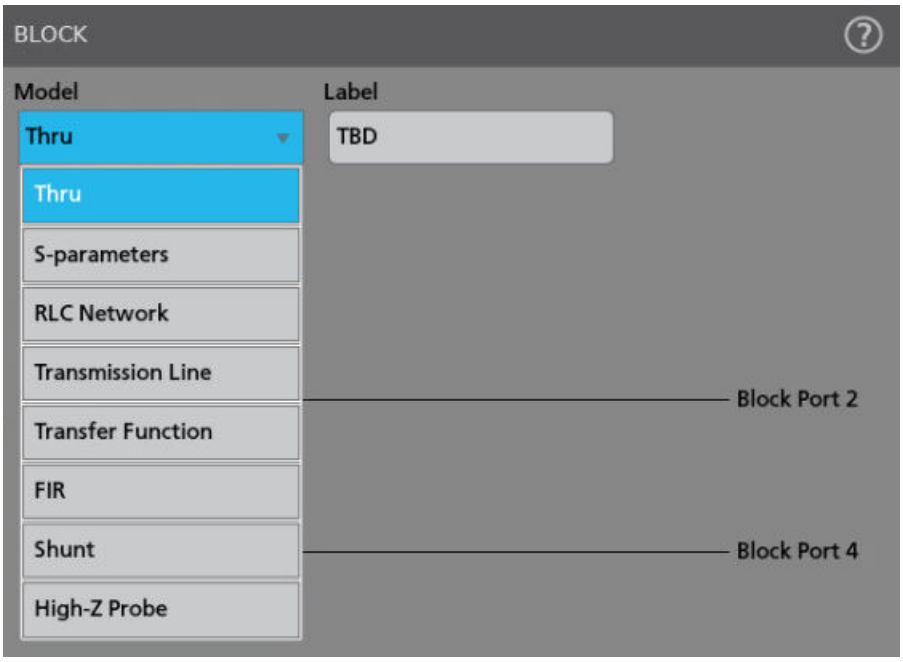
1. In the SIM, locate the block that you want to duplicate.
2. Right-click the block.
3. Tap **Duplicate**.
4. A new block appears above the selected block.
5. Select a drop point to place the duplicated block.
6. Configure the duplicated block as required.

NOTE: Duplicate is available only for user-defined blocks. Receiver blocks cannot be duplicated.

Model Block menu

The model block contains the circuit models that represent the actual hardware probe, fixtures, etc. that were used to acquire the waveforms with the oscilloscope acquisition system. Here, you can define the effects of the fixture, probe, scope and other acquisition and measurement hardware upon the DUT signal, re-normalize the S-parameters reference impedance, perform singled-ended to mixed mode conversion, reach the Block Configuration menu for Thru, File, RLC and T-line options, add and configure High Z, SMA probes, or SAP, and many other tasks.

Generic blocks can be described as one of the following:



Thru: The Thru block model is used when the block is not a necessary part of the cascade. It represents an ideal model that has no effect upon signals in the system.

S-parameters: Supported S-parameters file types for the blocks include: 2, 4, 6, 8, 12, and 16-port, in addition to 4-port differential mixed mode. When S-parameters file is loaded, it plots the selected S-parameters. All S-parameters terms in the data are taken into account when computing test point transfer functions.

NOTE: When you generate your S-parameters file, do not put ';' delimiters at the end of each data line. It is incorrect format for touchstone file and may lead to improper reading when loading the file into SIM.

RLC Network: This allows RLC (resistor, inductor, and capacitor) elements to be used to model the block. SIM computes a set of 4-port S-parameters that will be used in the cascade. The cross-coupling terms will be set to zero.

Transmission Line: This allows you to define a lossless transmission line. SIM computes a set of 4-port S-parameters depending on the parameter settings. The cross-coupling terms shall be set to zero.

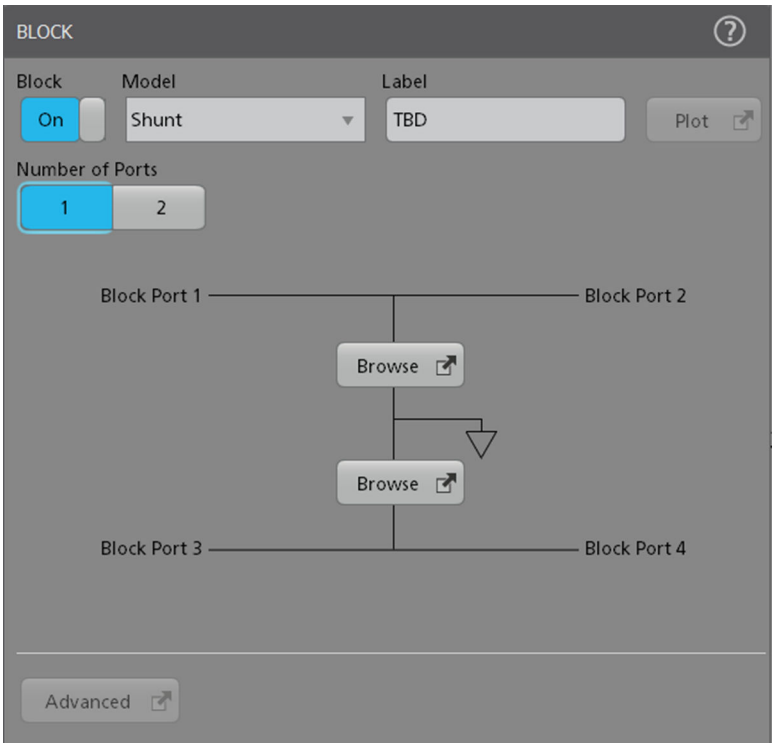
Transfer Function: Models a frequency domain set of complex data. This allows two files containing transfer function data in the frequency domain to be loaded to represent the block. The file will be in the s1p Touchstone 1.0 format. This format contains a frequency column, a real or magnitude column and an imaginary or phase column. SIM converts these into a single 4-port S-parameters set with ideal cross-coupling and reflection coefficient terms set to zero. A common use for this choice would be to represent a pair of shielded cables connected between a fixture and the oscilloscope.

FIR: Models a time domain impulse response. This allows two files containing FIR filter coefficients data in the time domain to be loaded to represent the block. The file shall be in the oscilloscope arbflt() ASCII format. SIM converts these into one 4-port S-parameters set with ideal cross-coupling terms set to zero. The reflection coefficient terms are also set to zero.

NOTE: When you generate your own .flt file, the keyword [DELAY] can only be specified before filter data. Placing this keyword after the filter data, results in 'delay not to be read in' and produce the wrong simulation results.

Shunt: Shunt connection is available with 1-Port Shunt and 2-Port Shunt.

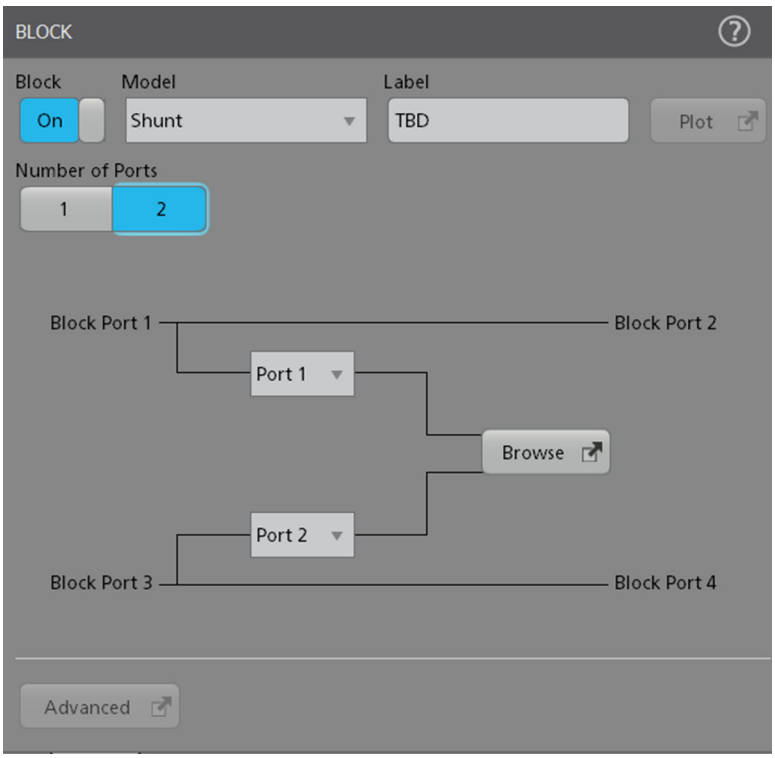
- **1-Port Shunt**: Models two 1-port S-parameters sets in shunt connection. This allows two 1-port S-parameters files to be loaded as a shunt connection.



SIM converts these into one 4-port S-parameters set in the cascade of blocks. A common use for this choice would be there are shunt connections represented by two 1-port S-parameters files.

- **2-Port Shunt**: Models one 2-port S-parameters set in shunt connection. This allows one 2-port S-parameters file to be loaded as a shunt connection.

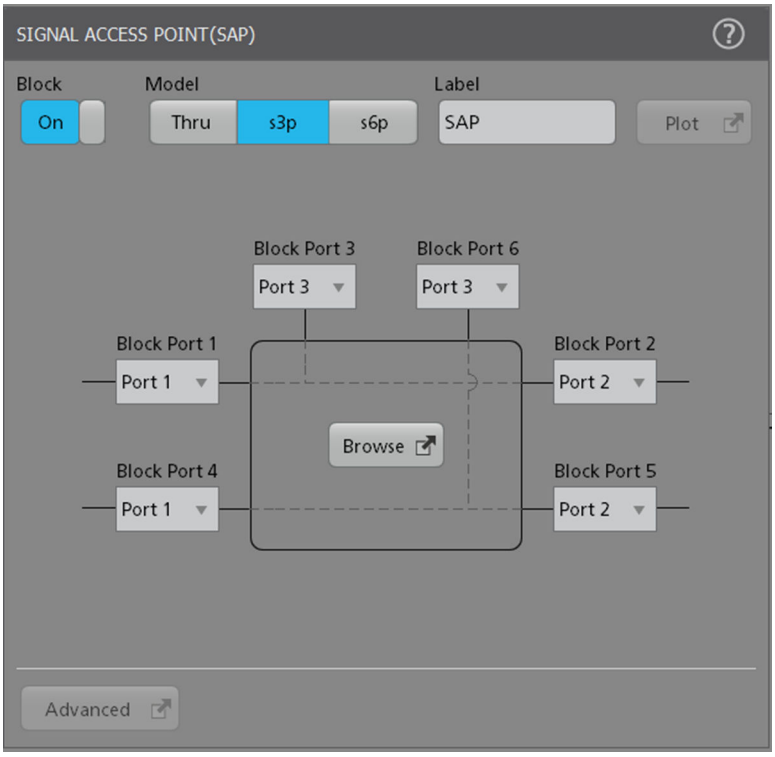
2-Port Shunt



SIM converts these into one 4-port S-parameters set in the cascade of blocks. A common use for this choice would be that there are shunt connections represented by a 2-port S-parameters file.

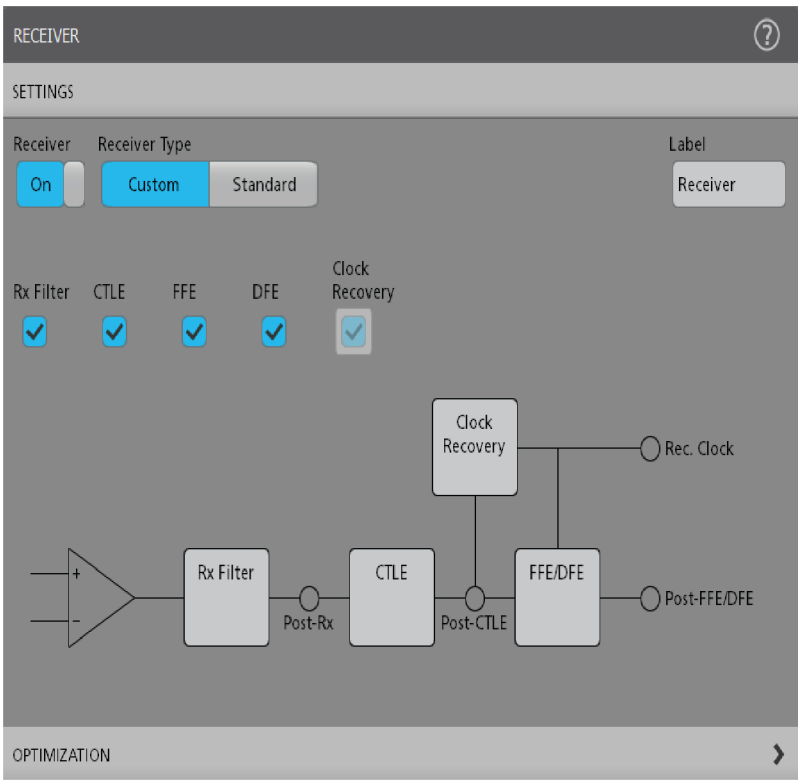
SAP

SIM can de-embed the effects of Signal Access Point (SAP) used for DDR. By using 3-port and 6-port S-parameters files, the effects of the SAP and probe on both the single-ended and differential signals can be removed. This model can also remove the effects of other measurement circuit blocks when probes are being used, for example, an RF switch.



Receiver Block

Models either a custom or a standard receiver. The configurable blocks available inside this block are Rx Filter, CTLE, Clock Recovery and FFE/DFE. When type is standard these blocks are configured to the values as defined by the values as defined by the selected standard.



See Clock Recovery configuration menu

See RX Filter configuration menu

See CTLE configuration menu

See FFE/DFE configuration menu

Test points

A test point can either have a single waveform or up to four waveforms associated with it. When only a single waveform is associated for each test point, test points appear as a green circle is placed on the dashed line between the two surrounding blocks. If more than one waveforms are associated for each test point, test points appear in one of two ways.

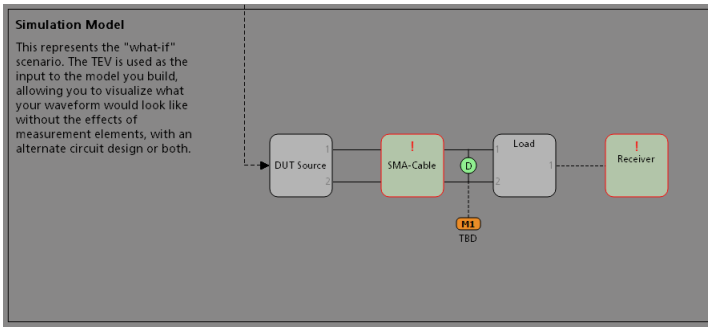
- When a single waveform is enabled, a letter represents which of the four possible waveforms is shown inside the test point and the test point is positioned appropriately.
- When more than one waveform is enabled, the letter represents the type of waveform is moved to the right of each math waveform test point.
  - D: Differential
  - C: Common Mode
  - A: Single-ended A leg
  - B: Single-ended B leg

The TEV test point is on left of the DUT Source block and can have up to four waveforms associated with it. This Test Point is available only in the Advanced workflow.

If **Source** is set to **None** in the model, you can create and save filter files for use elsewhere. But there are no math waveforms associated with test points.

Errors in blocks

If a one or more blocks contain errors or have incorrect or incomplete field, the block is highlighted with thick red border and a red exclamation icon (!) in the SIM configuration menu and the Calculate SIM button is grayed out.

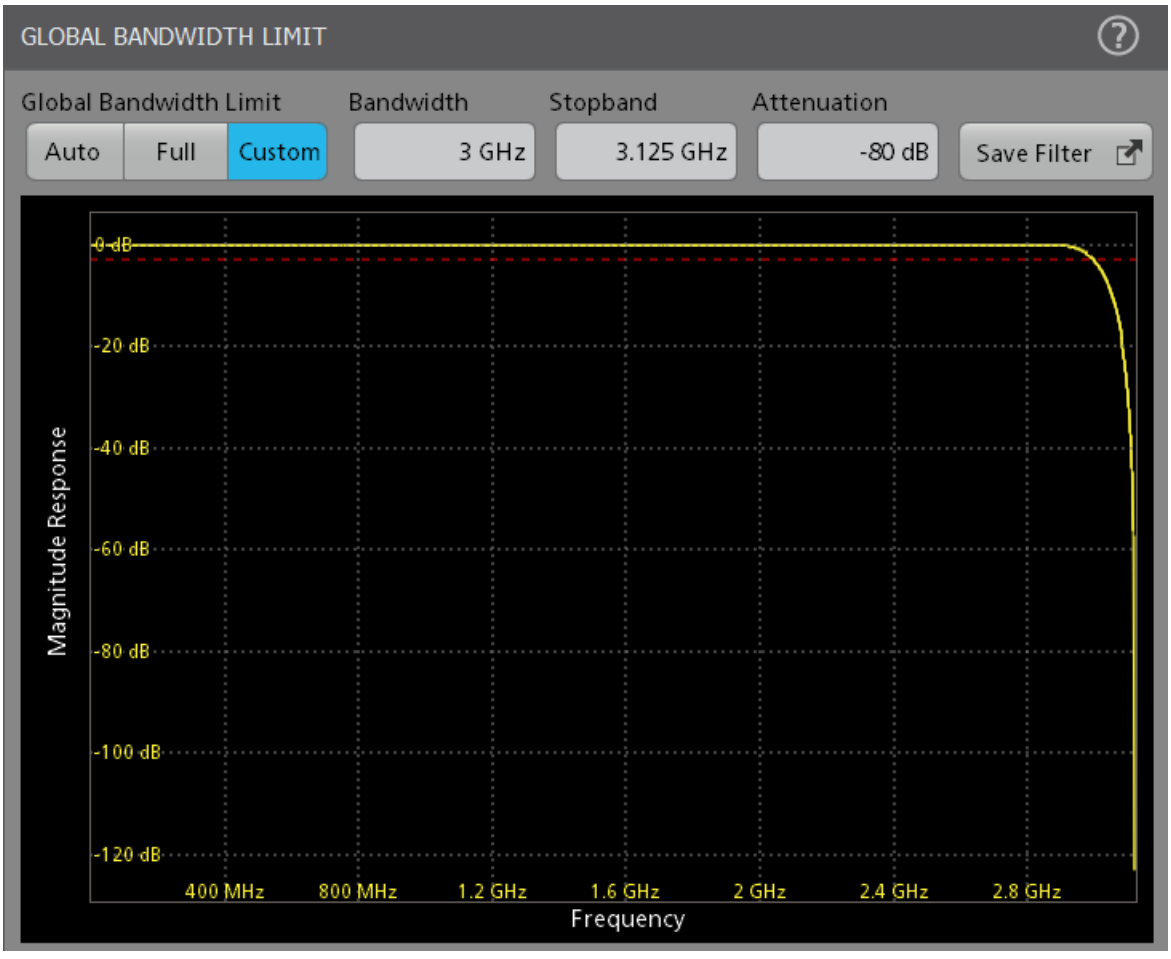


To view or resolve the error, double-tap on the block. Once all errors are corrected, the Calculate SIM button is enabled. This ensures that calculations are performed with a valid or complete data.

Refer to the SIM badge error and warning messages (on page 612) for more information.

Create a custom bandwidth limit filter

When de-embedding, it is critical to pay close attention to the signal to noise ratio at higher frequencies. While SIM can automatically determine the bandwidth limit for the de-embed filter, it is often necessary to fine tune the filter. SIM provides the ability to control the filter by setting the cutoff frequency with the Bandwidth control, and the transition bandwidth by setting the Stopband frequency, and the stopband attenuation in dB, using the Attenuation control.



Global Bandwidth Limit menu

Follow these steps to create a custom filter:

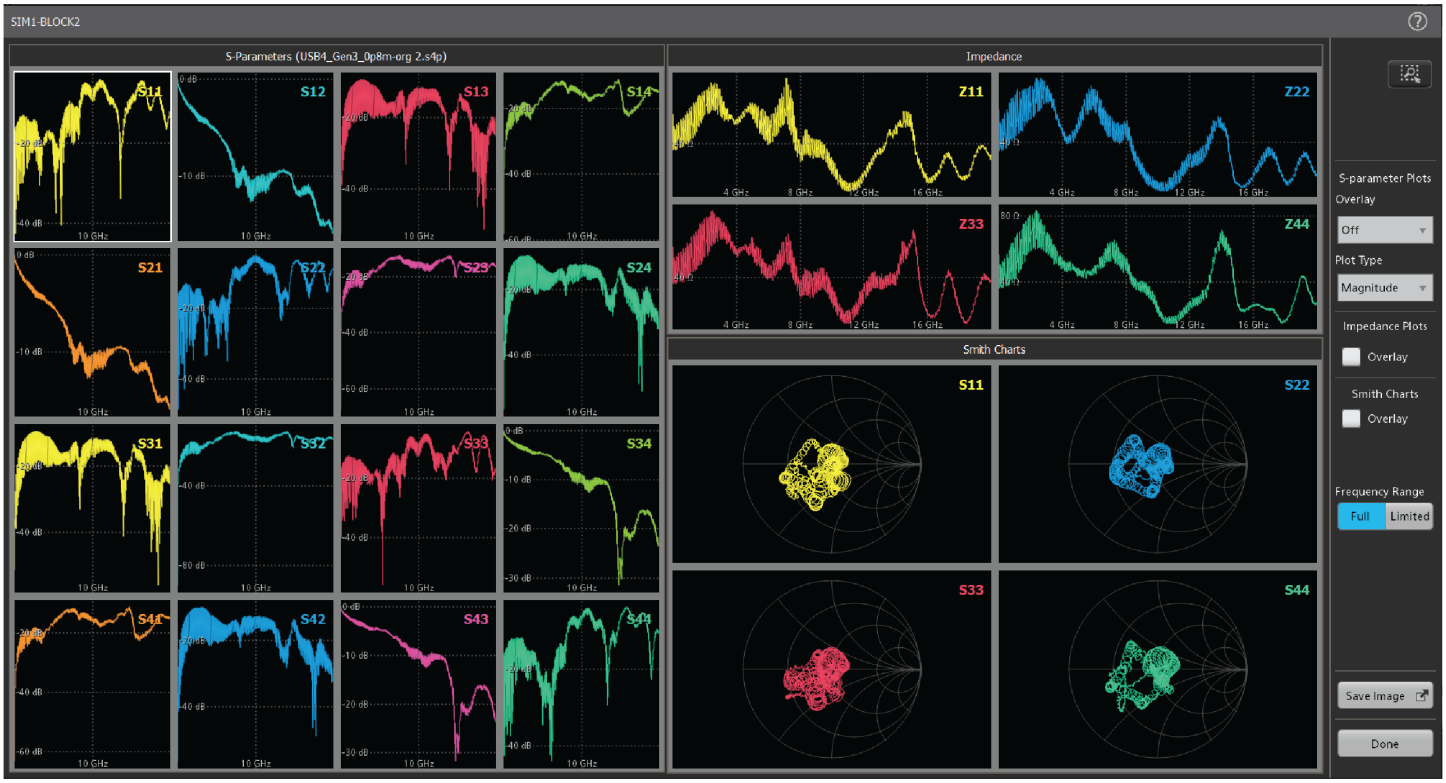
1. Tap **Global BW Limit** from the main menu.
2. Select Global Bandwidth Limit as **Auto**, **Full**, or **Custom**.
3. If **Custom** is selected, set the Bandwidth (GHz), Stopband (GHz) and Attenuation (dB) fields.
4. Tap anywhere outside the menu to close the menu.
5. Tap **Calculate SIM** and then **Plot Test Points**. The filter response is plotted.

S-parameter files often contain noisy or unreliable data at the highest measured frequencies. Limiting the bandwidth helps maintain waveform accuracy and stability during analysis.

The Global BW Limit sets a uniform bandwidth filter for all S-parameter blocks in the model, controlling the highest frequency content used in the simulation to improve stability, reduce noise, or meet specific test bandwidth requirements.

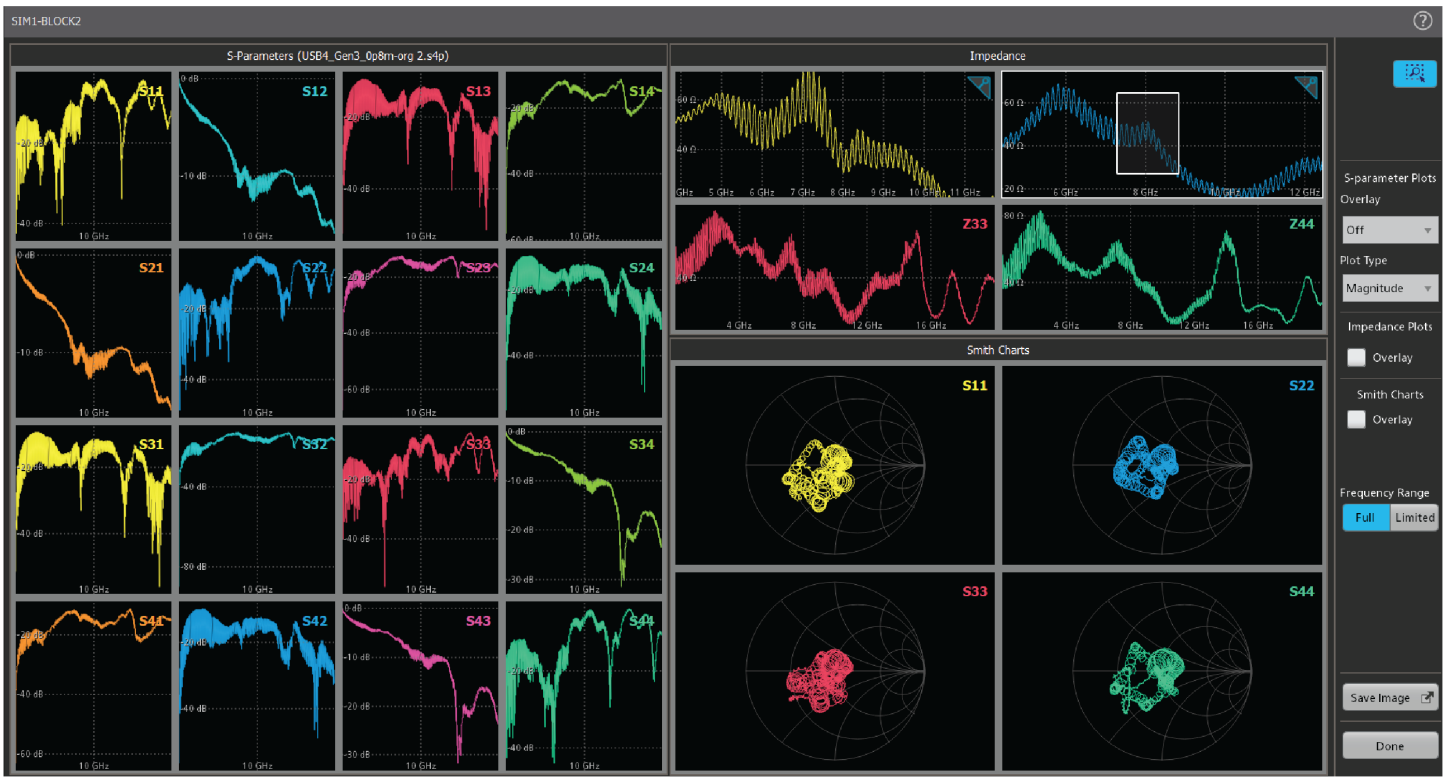
Plots

To support model validation and debugging, SIM provides extensive plotting capabilities for both S-parameters-blocks and test point filters. These include time and frequency domain magnitude plots, phase response, impulse and step responses, impedance vs. frequency, and Smith charts. The **Plots** window includes a zoom feature that enables detailed examination of specific regions within a plot. These visualizations help you to gain deeper insight into model behavior and confidently interpret test point results.



Single click on the Draw-A-Box Zoom icon (button in the upper-right corner) and drag to create a rectangular selection on any plot to zoom into the desired region. Double-click the Draw-A-Box Zoom icon to activate continuous zoom mode, which enables zooming across all plots in the plot window without needing to reselect the tool each time. This functionality is consistent across all SIM plot windows.

Additionally, double-clicking any plot (such as S11 or S12) opens it in a maximized view and automatically restores the Draw-A-Box Zoom icon to its default state.



Plot types include

Plots are available for each block in the De-embed and Embed blocks with Magnitude, Impulse Response, Step Response, and Phase plots for each test point.

- Magnitude vs. Frequency
- Phase vs. Frequency
- Impedance as a function of frequency
- **Impedance vs. Magnitude plot**: These plots let you quickly observe how impedance varies over frequency for the reflection coefficients in the S-parameters set.
- **Smith Chart plot**: A Smith Chart provides impedance, phase, and magnitude information, including impedance in polar or rectangular format, markers readout, start and stop frequencies, and overlay.
- **Impulse response and Step response**: In the time domain, all the plots should be settled by the end of the time record. If not, this could mean that the measurement was performed with too short of a time interval. Note that if a pulse is close to the beginning of the record then it should be expected that a portion of the non-causal response at the front is wrapped to the very end of the record. SIM handles the wrapped-end part internally.

  This non-causal part is not real for the analog circuit that was measured; it is an artifact of the band-limiting effect of performing the IFFT operation to transform the S-parameters set to the time domain.

  If the frequency spacing is too wide, the time domain shows aliasing where the pulse is wrapped in the time domain to an aliased position.

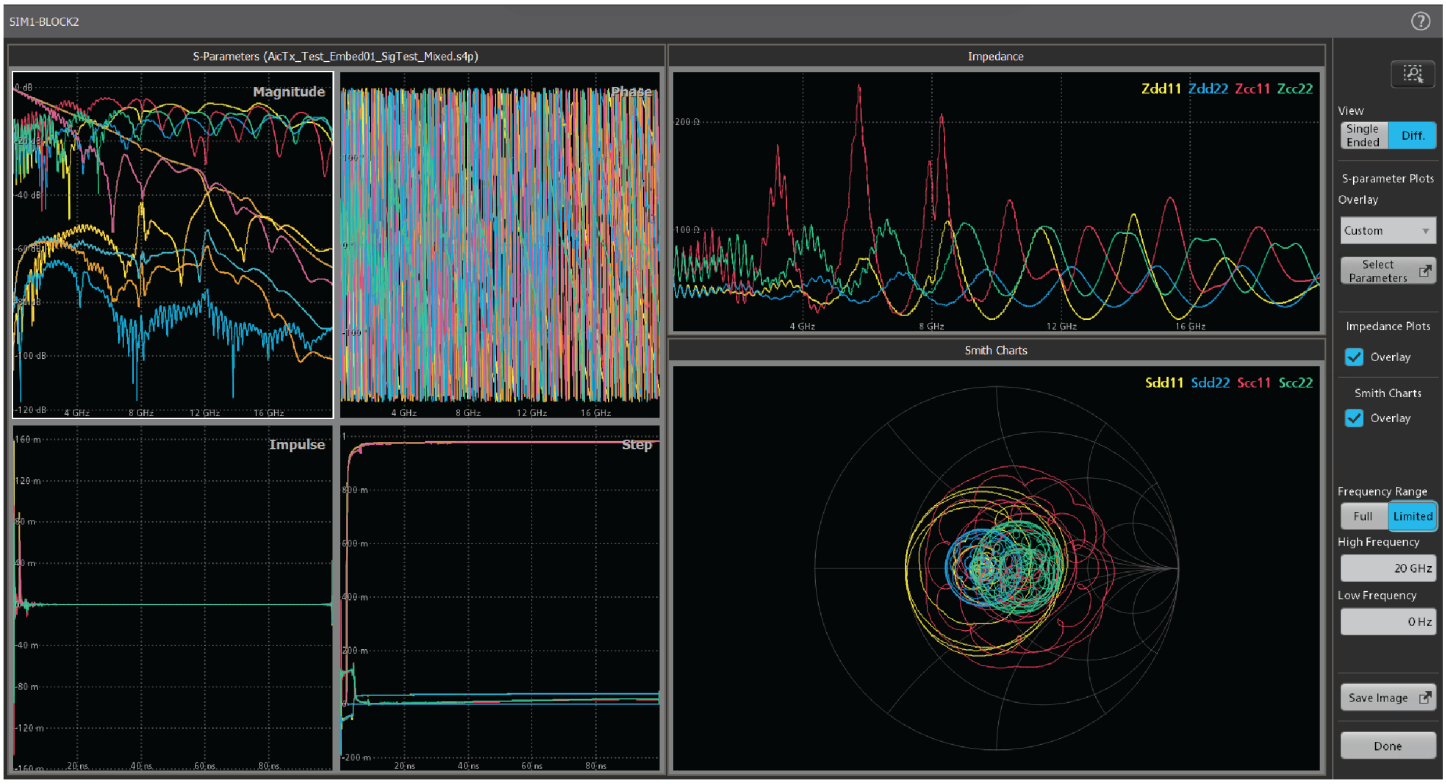
Use these plots to

- Quickly verify port assignment.
- Validate that the magnitude is correct. Typically, the insertion loss should be higher at high frequencies; reflection and crosstalk is typically higher at higher frequencies.
- Do a passivity check. None of the S-parameters of a passive system should go above 0 dB in the frequency domain. All data can be less than 0 dB and yet the system may still violate passivity where the total output power is more than the input power.

Using plots for troubleshooting S-parameters

SIM S-parameters plots include an Overlay option which allows you to view any selected plots on one display. This can be helpful in many scenarios, such as:

- **Viewing a DUT with Mismatched Differential Pairs**: In this example, impulse plots reveal any differences in delay between the two cables. In the overlay magnitude plot overlay is showing the difference in attenuation between the two cables.



- **Verifying mixed mode vs. Single-ended mode**: You can use these plots to verify the difference between mixed mode and single-ended.
- **Troubleshooting bad step response**: A step response may have a non-casual dip before the rising edge. The cause of this may be revealed by observing the magnitude response zoomed in at DC. The S-parameters commonly do not contain DC, because VNAs cannot measure it. SIM must then extrapolate the data to DC before it can be transformed to the time domain for processing to make a transfer function filter. Extrapolation can be problematic, where the first data sample in the S-parameters set may be lower than the previous one due to noise or a bad VNA measurement. This results in the extrapolated portion of the curve being slightly offset by a few tenths of a dB. This is enough to cause the pre-shoot dip in the step response. A 2 degree phase error at frequencies around DC may also cause issues like this.
- **Troubleshooting bad VNA measurements:** The plots may be used to determine if S-parameters set measured on a VNA or obtained from a circuit simulator, have issues. For example, it is relatively common that someone does not properly torque the cable connections during the VNA measurement process. This can result in frequency dip anomalies in the S-parameters frequency response. This commonly happens if the connectors were tightened by hand without using a torque wrench.

Examples of tasks and troubleshooting

This oscilloscope has the optional Signal Integrity Modeling (SIM) package that allows you to build a model of your system under test, build a model of a simulated system the DUT will operate in, and view the signal at any point in the system. Using SIM, you can acquire the signal where you can, view the signal where you want.

Here are some examples of the many tasks that SIM can perform:

- De-embedding the cables (on page 136)
- Embedding the cables (on page 137)
- De-embedding and Embedding the cables using Advanced Workflow (on page 138)
- Stub de-embedding simulation (on page 143)
- Stub embedding simulation (on page 140)

De-embedding the cables

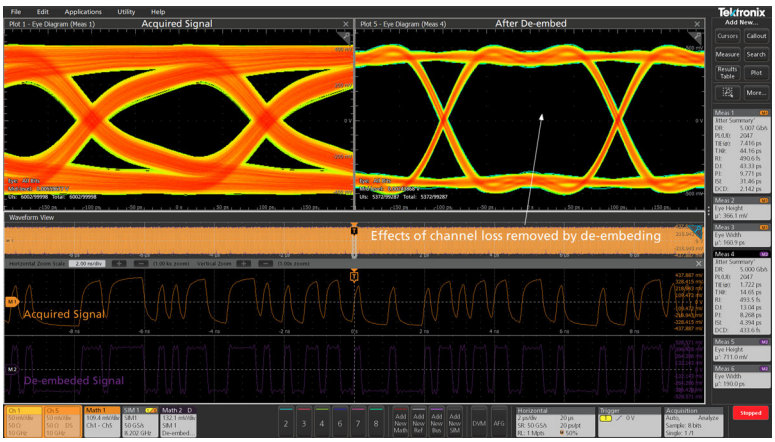
- In high-speed serial systems, it is common for the signal being transmitted to experience significant amounts of loss and inter-symbol interference (ISI) due to channel effects. When evaluating the performance of a high-speed serial transmitter, it is typically impractical to acquire the signal directly at the TX pins of the device under test (DUT) and instead the signal must be accessed at the end of test channel, where it has already been impaired by channel effects.
- Using the Simple De-embed Workflow, you can quickly build a model of the channel between your DUT and the oscilloscope and then view the signal at the TX pins without the effects of the channel and oscilloscope attached to the DUT. In this workflow, the channel and oscilloscope are replaced with an ideal load.
- In this demo session, the signal acquired is a PRBS11 pattern NRZ signal operating at 5.0 Gbps. The channel is a 24-inch differential PCB trace and a pair of 1 meter phase matched cables.
- Below setup includes the Eye Diagrams and Eye Height and Eye Width measurements for the original signal as acquired by the oscilloscope as well as the signal after de-embedding the effects of the channel.

Procedure

1. Attach a probe to channel 1 of the oscilloscope and the positive signal of the differential signal at the end of a 24” PCB trace.
2. Attach a probe to channel 3 of the oscilloscope to the negative signal of the differential signal at the end of a 24” PCB trace.

   NOTE: For the best results, use the two channels that provide the highest real-time sample rate while operating concurrently.

3. Double-tap the **Horizontal** badge in the Settings bar and select **Horizontal Mode** > **Manual** and set **Sample Rate** to maximum value (no interpolation IT mode) and **Record Length** to approximately 1M.
4. Tap **Add New Math** and set the function as CH1 – CH3.
5. Tap **Add New Measurement** > **Jitter** > **Jitter Summary** > **Add**.
6. Double-tap the measurement badge, select **CONFIGURE** and set the **Source 1** to **Math 1**.
7. Tap **Add New Measurement** > **Jitter** > **EYE MEASUREMENTS** > **Eye Height** > **Add**.
8. Double-tap the measurement badge, select **CONFIGURE** and set the **Source 1** to **Math 1**.
9. Similarly, add **Eye Width** measurement and set the **Source 1** as **Math 1**.
10. Tap **Add New SIM** in the Settings bar.
11. Select the **Workflow** as **Simple De-embed**.
12. Design a **Physical Model** as per requirement.
13. In the Simple De-embed Workflow, the Simulation Model represents the DUT connected to an ideal load and a single Test Point is provided to view the signal coming from the DUT. Set the De-embed Test Point output on **Math 2**.
14. Tap **Calculate SIM** to build the output math filters.
15. Add **Jitter Summary**, **Eye Height**, and **Eye Width** measurements and set the **Source 1** to Math 2.



Embedding the cables

- In high-speed serial systems, it is common for the signal being transmitted to experience significant amounts of loss and inter-symbol interference (ISI) due to channel effects. When the signal reaches the far end of the channel, it will often be highly attenuated and distorted, shrinking the eye seen at the receiver, even closing it completely.
- To evaluate the ability of a high-speed serial transmitter to provide an open eye to the receiver at the far end of the channel, the signal needs to be put through a channel and acquired at the far end.
- Often these high-speed serial transmitters will be integrated into many different systems. To ensure that they will perform correctly, they need to be tested in many different systems.
- Building physical hardware to evaluate the performance of high-speed serial transmitters in many different systems takes a lot of time, is expensive and usually deemed impractical. Instead, it is usually better to build an evaluation board for the device under test (DUT), acquire the signals from this board and then simulate the effects of various channels on the signal.
- Using the Simple Embed workflow, you can quickly build a model of the channel to be simulated and then view the signal from your DUT at the far end of that channel. This workflow assumes that the oscilloscope is directly connected to the DUT transmitter and the signal loss between the DUT and oscilloscope is negligible.
- In the Simple Embed Workflow, the Physical Model represents the DUT connected directly connected to the oscilloscope. The system to be simulated is modeled in the Simulation Model. A single Test Point is included by default and is placed at the far end of the simulated channel.
- This setup includes Eye Diagrams and Eye Height and Eye Width measurements for the original signal as acquired by the oscilloscope as well as the signal after embedding the effects of the channel.
- In this demo session, the signal acquired is a PRBS11 pattern NRZ signal operating at 5.0 Gbps. The channel being simulated is a 40-inch differential PCB trace.

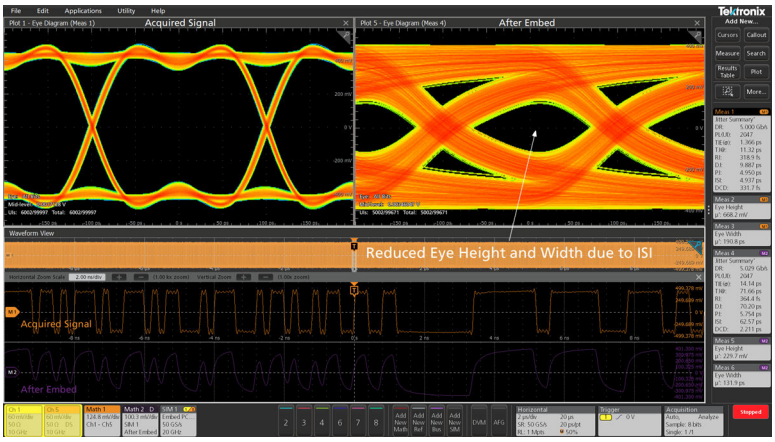
Procedure

1. Attach a probe to channel 1 of the oscilloscope and the positive signal of the differential signal at the DUT transmitter.
2. Attach a probe to channel 3 of the oscilloscope to the negative signal of the differential signal at the DUT transmitter.

   NOTE: For the best results, use the two channels that provide the highest real-time sample rate while operating concurrently.

3. Double-tap the **Horizontal** badge in the Settings bar and select **Horizontal Mode** > **Manual** and set **Sample Rate** to maximum value (no interpolation IT mode) and **Record Length** to approximately 1M.
4. Tap **Add New Math** and set the function as CH1 - CH3.
5. Tap **Add New Measurement** > **Jitter** > **Jitter Summary** > **Add**.
6. Double-tap the measurement badge, select **CONFIGURE** and set the **Source 1** to **Math 1**.
7. Tap **Add New Measurement** > **Jitter** > **EYE MEASUREMENTS** > **Eye Height** > **Add**.

8. Double-tap the measurement badge, select **CONFIGURE** and set the **Source 1** to **Math 1**.
9. Similarly, add **Eye Width** measurement and set the **Source 1** as **Math 1** to the measurement.
10. Tap **Add New SIM** in the Settings bar.
11. Select the **Workflow** as **Simple Embed**.
12. Design a **Simulation Model** as per requirement.
13. Set the simulated embed Test Point at the far end of the channel output on **Math 2**.
14. Tap **Calculate SIM** to build the output math filters.
15. Add **Jitter Summary**, **Eye Height** and **Eye Width** measurements and set the **Source 1** to **Math 2**.



De-embedding and Embedding the cables using Advanced workflow

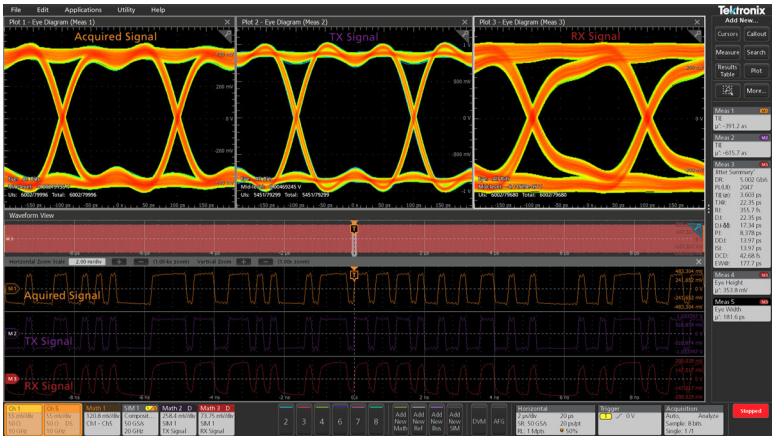
- When evaluating the performance of a high-speed serial transmitter, it is typically impractical to acquire the signal directly at the TX pins of the device under test (DUT) and instead the signal must be accessed at the end of test channel, where it has already been impaired by channel effects.
- Using the Advanced workflow, you can build a model of the channel between your DUT and the oscilloscope as well as model the characteristics of your DUT transmitter using the Physical Model. In the Simulation Model, you can build a model of the system in which your DUT will operate as well as modify the characteristics of your DUT transmitter and the load of the receiver at the far end of the channel.
- As many Test Points as needed can be added to both the Physical and Simulation Models to view the signal at any point in composite model, including the signal of the Thevenin Equivalent Voltage source that represents the ideal transmitter.
- The views in this demo were created by adding a new SIM and building a composite system model using the Advanced workflow. The system used to acquire the signal, including the oscilloscope, is modeled in the Physical Model. The target system in which the DUT will operate is modeled in the Simulation Model.
- This setup includes Eye Diagrams for the signal as acquired by the oscilloscope, the Thevenin Equivalent Voltage source representing the ideal transmitter, and the signal at the far end of the simulated channel. Jitter Summary, Eye Width, and Eye Height measurements are also included for the far end signal.
- In this demo session, the signal acquired is a PRBS11 pattern NRZ signal operating at 5.0 Gbps.
- The physical system in which the DUT signal was acquired is a channel is a 2.42-inch differential PCB trace and a pair of 1 meter phase matched cables.
- The target system being simulated from the DUT transmitter to the host receiver is 2.42-inch PCB trace to a card edge connector to a 12-inch PCB trace to the host receiver.

Procedure

1. Attach a probe to channel 1 of the oscilloscope and the positive signal of the differential signal at the DUT transmitter.
2. Attach a probe to channel 3 of the oscilloscope to the negative signal of the differential signal at the DUT transmitter.

NOTE: For the best results, use the two channels that provide the highest real-time sample rate while operating concurrently.

3. Double-tap the **Horizontal** badge in the Settings bar and select **Horizontal Mode** > **Manual** and set **Sample Rate** to maximum value (no interpolation IT mode) and **Record Length** to approximately 1M.
4. Tap **Add New Math** and set the function as CH1 - CH3.
5. Tap **Add New Measurement > Jitter > Jitter Summary** > **Add**.
6. Double-tap the measurement badge, select **CONFIGURE** and set the **Source 1** to **Math 1**.
7. Tap **Add New Measurement** > **Jitter** > **EYE MEASUREMENTS** > **Eye Height** > **Add**.
8. Double-tap the measurement badge, select **CONFIGURE** and set the **Source 1** to **Math 1**.
9. Similarly, add **Eye Width** measurement and set the **Source 1** as **Math 1** to the measurement.
10. Tap **Add New SIM** in the Settings bar.
11. Select the **Workflow** as **Simple Embed**.
12. Design a **Simulation Model** as per requirement.
13. Set the simulated embed Test Point at the far end of the channel output on **Math 2**.
14. Tap **Plot** > **Eye Diagram** > **Add** on the Results bar.
15. Double-tap the plot badge, select **CONFIGURE** and set the **Source 1** to **Math 1**.
16. Select the **Workflow** as **Advanced** in the SIM.
17. Design a **Simulation Model** as per requirement.
18. Tap **Add Test Point** as **Test Point M2** and place it at the Thevenin Equivalent Voltage Source Test Point to the left of the Source Block in the Simulation Model.
19. Tap **Add Test Point** as **Test Point M3** and place it at the far end of the channel, just left of the Load block.
20. Tap **Calculate SIM** to build the output math filters.
21. Add **Eye Diagram** plot on Results bar and set the **Source 1** as **Math 2**.
22. Add **Jitter Summary**, **Eye Height**, and **Eye Width** measurements and set the **Source 1** to **Math 3**.

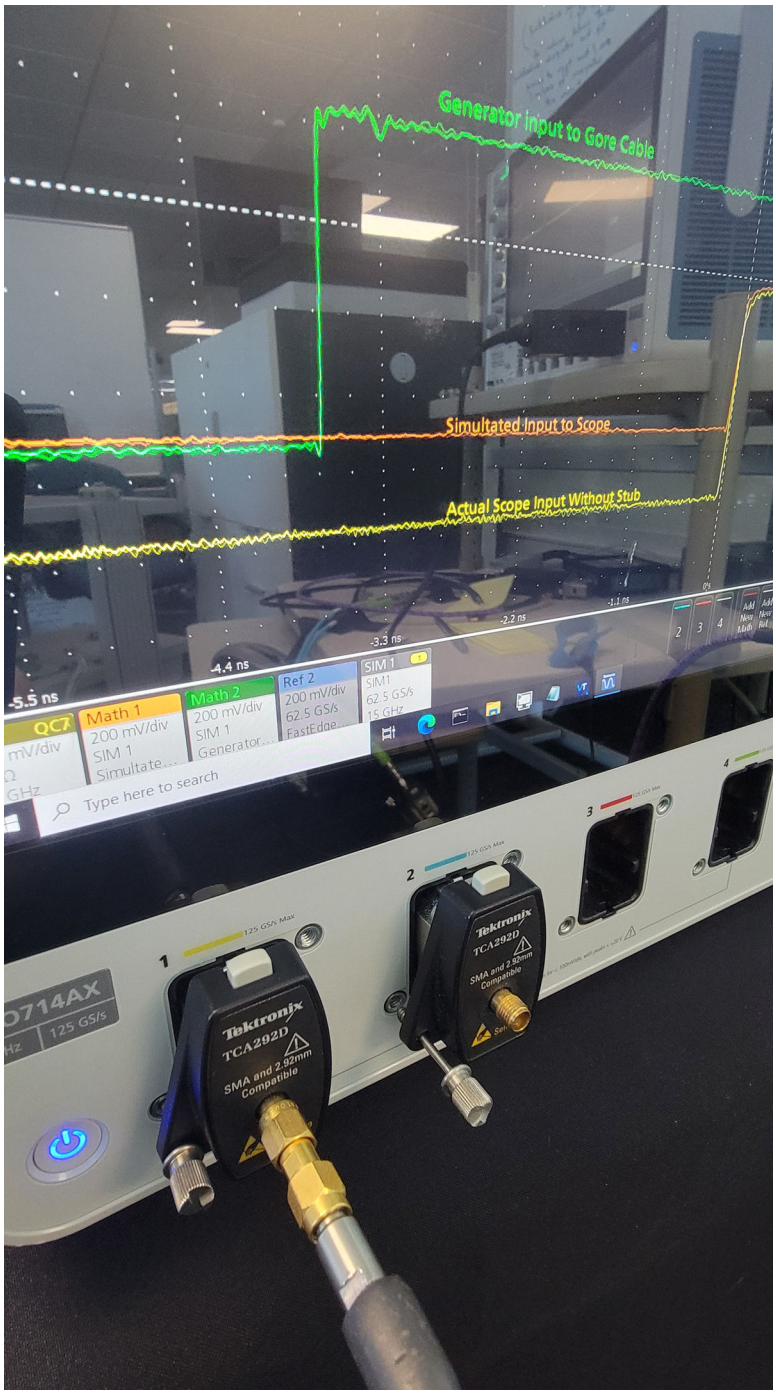


Stub embedding simulation

SIM can acquire a waveform and generate simulated waveforms based on the user defined model configuration. The following example demonstrates acquiring the signal without the stub and simulating the effect of embedding the stub. By applying the filters to the acquired waveform, SIM creates a representation of what the signal would look like if the stub were present.

Procedure

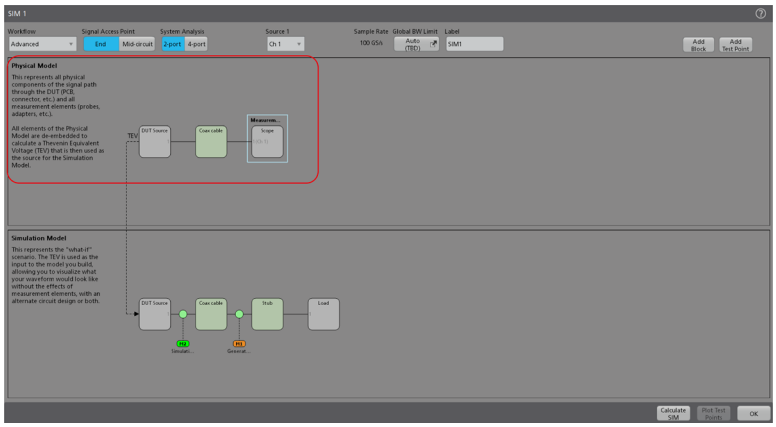
1. Attach a coax cable to channel 1 of the oscilloscope with no stub is connected to it.
2. Connect the other end of the cable to the Fast Edge output of the oscilloscope. The oscilloscope acquires SIM physical circuit waveform into Ch1.



3. Tap **Add New SIM** in the Settings bar.

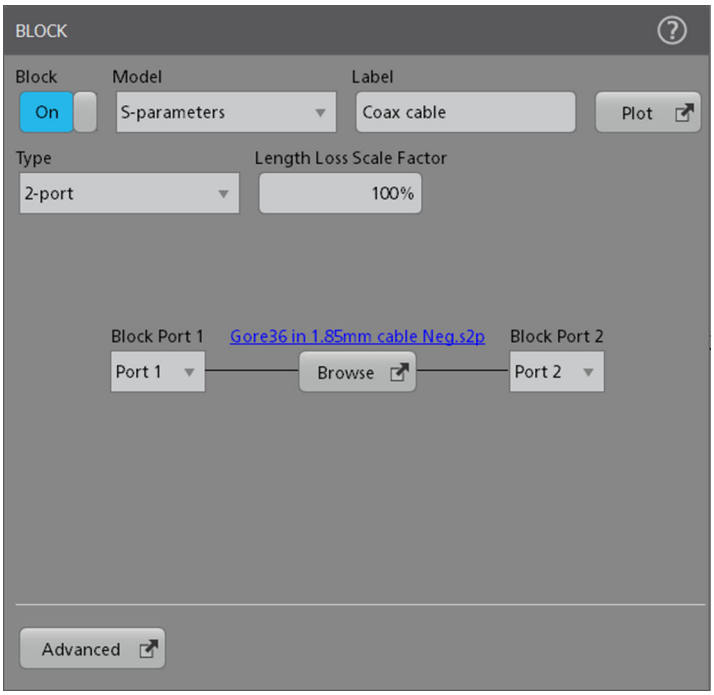
4. Select the **Workflow** as **Advanced**.
5. To design a **Physical Model** follow below steps.

Stub embedding physical model



a. Double-tap the **DUT Source** and set the **Model** as **Nominal** (to 50 ohm default value of source impedance). Tap outside to close the menu.

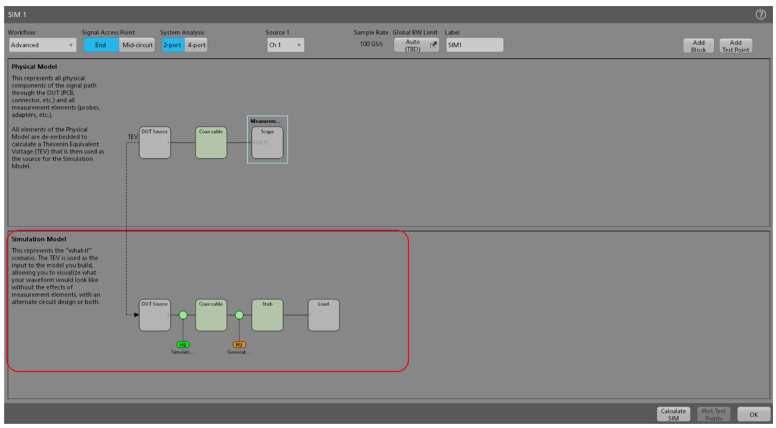
b. Double-tap the **TBD (Thru)** and configure it as follows:



- Set **Model** as **S-parameters**.
- Double-tap and enter the **Label** as **Coax cable**.
- Set the **Type** as **2-port**.
- Tap on **Browse** and select the S-parameters file.

c. Double-tap the **Scope** and set the **Model** as **Nominal** (to 50 ohm default value of source impedance). Tap outside to close the menu.

6. To design a **Simulation Model** follow below steps.

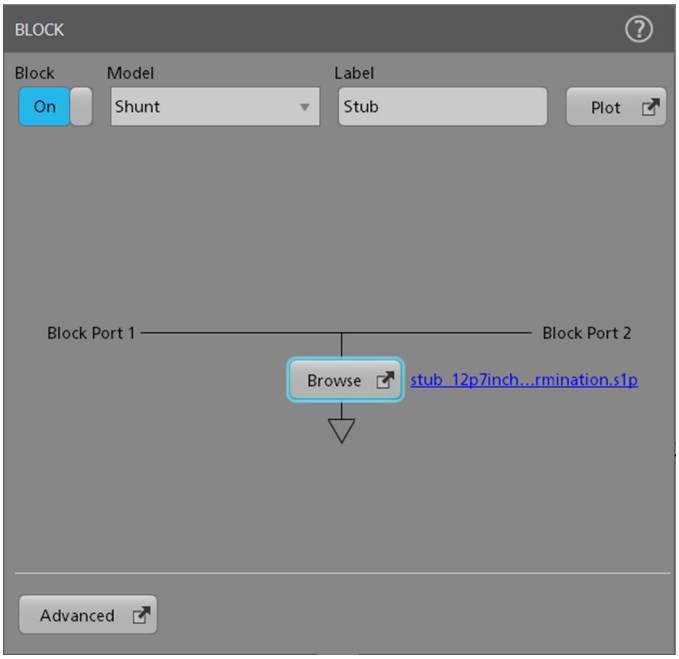


Stub embedding simulation model

a. Double-tap the **DUT Source** and set the **Model** as **Nominal** (to 50 ohm default value of source impedance). Tap outside to close the menu.

b. Double-tap the **TBD (Thru)** and configure it the same way as the Coax cable in the Physical Model.

c. Tap **Add Block**, select the block and place it after coax cable in the simulation model. Configure it as follows:



- Set **Model** as **Shunt**.
- Double-tap and enter the **Label** as **Stub**.
- Tap on **Browse** and select the S-parameters file.

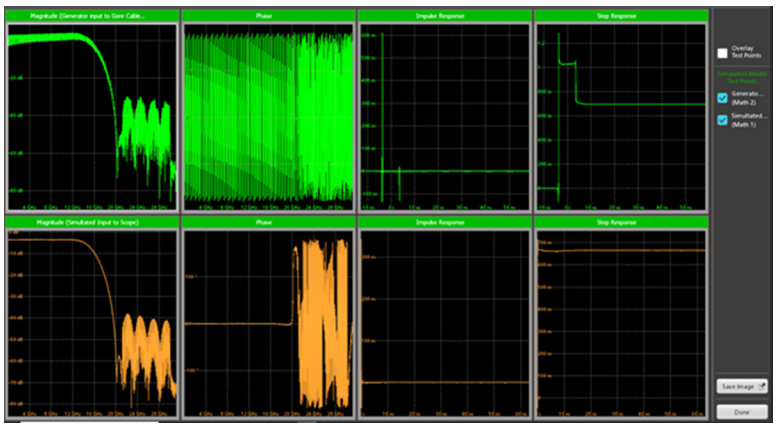
d. Double-tap the **Load** and set the **Model** as **Nominal** (to 50 ohm default value of source impedance). Tap outside to close the menu.

7. Tap **Add Test Point** as **Test Point M2** and place it at the left of coax cable block in the simulation model. The Test Point M1 is available by default at the left of load in the simulation model.

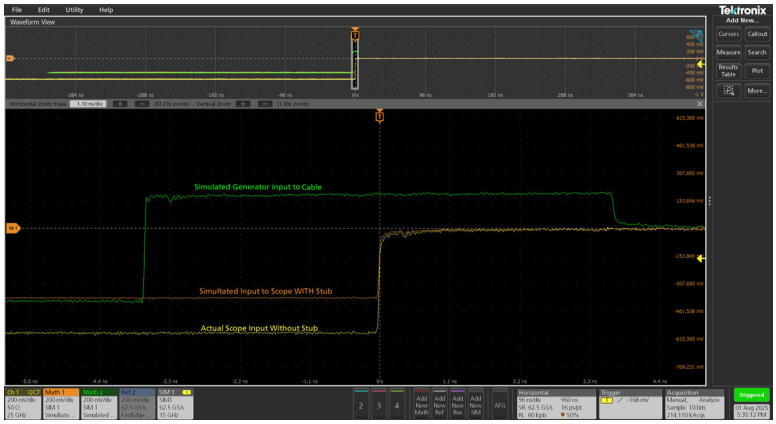
   Since the stub is short from port 1 to port 2 with a shunt to ground, the placement of test point M1 either before or after the stub block has no impact on the model behavior.

8. Tap **Calculate SIM** to build the output math filters. These filters are applied to the acquired input waveform on channel 1 in order to produce the live simulated waveform at the input to the cable. These live simulated waveforms at the input to the oscilloscope are simulating what would be seen if the stub is embedded into the simulation circuit.

9. Tap **Plot Test Points**. Observe the test point filter plots, the filters appear to be well behaved with characteristics that would be representative of simulating the reflections in the resulting waveforms.



The results are displayed on the oscilloscope screen below:



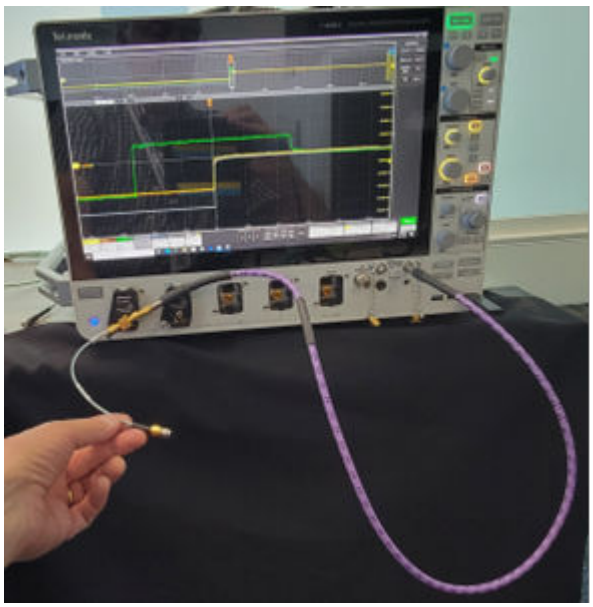
- The green waveform represents the signal at the output of the step generator, entering the coax cable. Notice that the step is vertically offset due to the magnitude of the reflection returning from the stub.
- After the signal's time delay traveling through the cable, the orange waveform shows the step at the oscilloscope input with the shunt stub in place.
- The orange waveform has a lower amplitude than the yellow incident waveform (which physically appears at the oscilloscope input). This reduction occurs because the combined input impedance of the stub and the oscilloscope is 25 ohms. Since this impedance is less than the 50-ohm impedance of the cable, the reflected step is inverted and subtracts from the incident step.
- Once the reflection travels back to the generator, the green waveform also shows the subtraction caused by the reflection, resulting in another smaller step at that time.

Stub de-embedding simulation

This example demonstrates how to use the SIM to model a cable and stub setup at the oscilloscope input, where a short stub coax cable is terminated with 50 ohms. Setup the model blocks to de-embed the Stub but leave the cable and show the simulated waveform at test points at each end of the cable.

Procedure

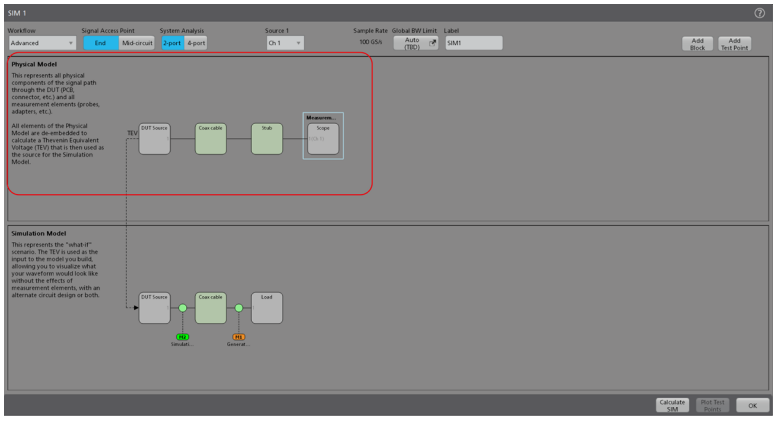
1. Connect a coax cable the fast edge output to the oscilloscope channel 1.
2. Connect the other end of the coax cable to a T connector that has its junction connected to the input of the oscilloscope.
3. Connect the other side of the T connector to the stub.



The stub is impedance matched across all frequencies, resulting in no reflections. Consequently, it appears as a 50 ohm input in parallel with the oscilloscope's 50 ohm input, effectively terminating the coax cable with 25 ohms. This creates a reflection back to the fast edge generator connected to the opposite end of the coax cable.

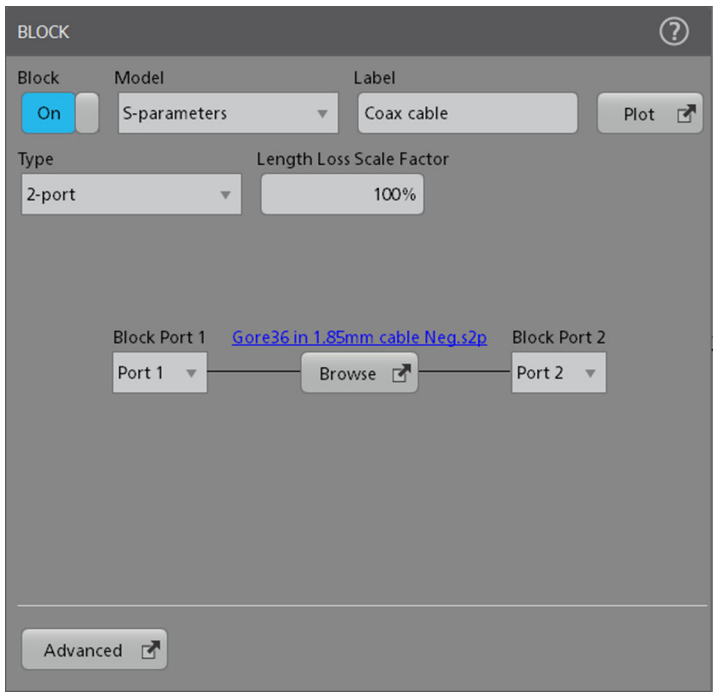
4. Double-tap the **Horizontal** badge in the Settings bar and select **Horizontal Mode > Manual**.
5. Set **Sample Rate** to as low as possible and **Record Length** to greater than 50k.

   Setting the sample rate to low still include the bandwidth of the signal of interest without aliasing harmonics. If sample rate is set too high it may result in the computed test point filters not being long enough to cover the desired time interval needed for a correct result.
6. Tap **Add New SIM** in the Settings bar.
7. Select the **Workflow** as **Advanced**.
8. Set the **Signal Access Point** to **End**.
9. Set the **System Analysis** to **2-port**.
10. Select **Source 1** as **Ch1**.
11. Tap **Global BW Limit** and select **Custom** and enter **Bandwidth** as **8 GHz**.
12. To design a **Physical Model** follow below steps.



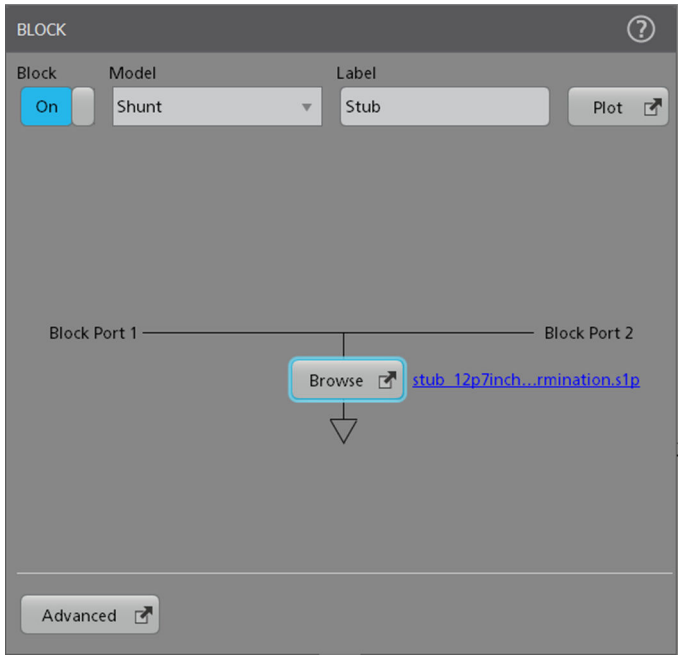
Stub de-embedding physical model

a. Double-tap the **DUT Source** and set the **Model** as **Nominal** (to 50 ohm default value of source impedance). Tap outside to close the menu.
b. Double-tap the **TBD (Thru)** and configure it as follows:



- Set **Model** as **S-parameters**.
- Tap and enter the **Label** as **Coax cable**.
- Set the **Type** as **2-port**.
- Tap on **Browse** and select the S-parameters file.
- Set the Block Port(s) to match which direction the cable is physically connected in the circuit.
- Tap **Plot** to observe the various plots representing the cable model.

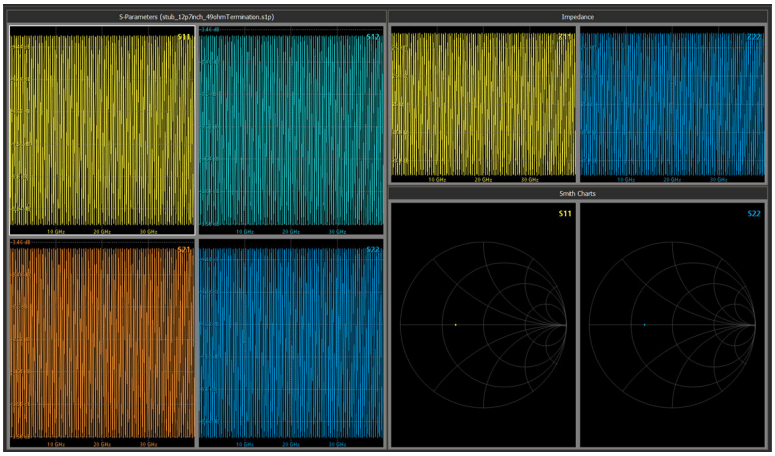
c. Tap **Add Block**, select the block and place it after coax cable block in the physical model. Configure it as follows:



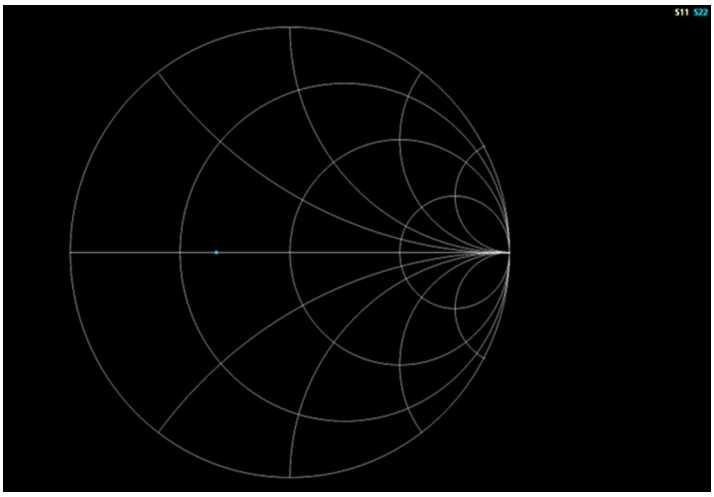
- Set **Model** as **Shunt**.
- Double-tap and enter the **Label** as **Stub**.
- Tap on **Browse** and select the S-parameters file.

d. Double-tap the **Scope** and set the **Model** as **Nominal** (to 50 ohm default value of source impedance). Tap outside to close the menu.

e. Tap the **Plot** to see the characteristics of the STUB model by plotting the S-parameters in various formats.

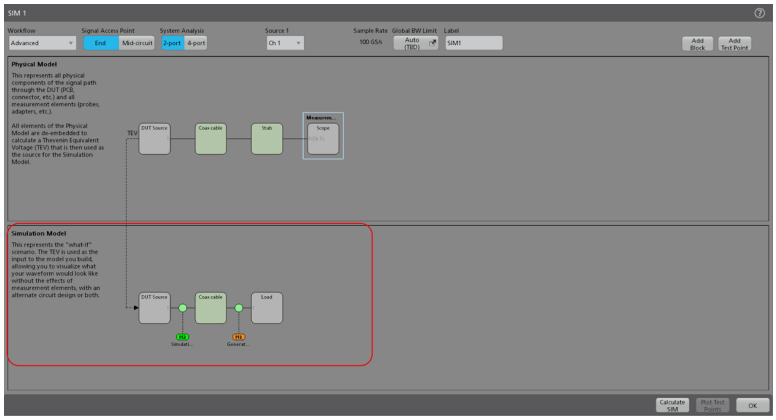


f. Double-tap on the Smith chart to expand it into a separate window, and observe that S11 at all frequencies is sitting on the resistive line at 25 ohms.



This represents the shunt 50 ohm stub in parallel with the 50 reference impedance at the port 2 of the two port block model.

13. To design a **Simulation Model** follow below steps.



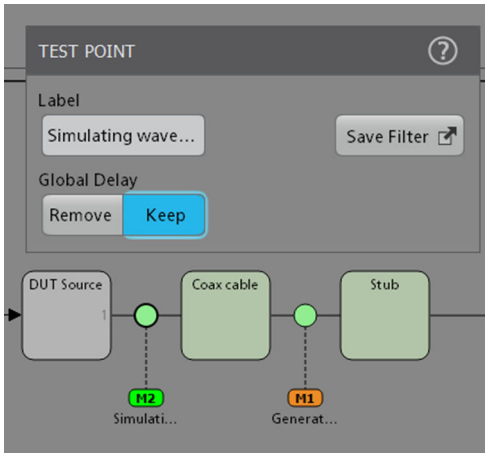
Stub de-embedding simulation model

a. Configure the DUT Source and coax cable as configured in the Physical Model.

b. Double-tap the **Load** and set the **Model** as **Nominal** (to 50 ohm default value of source impedance). Tap outside to close the menu.

14. Tap **Add Test Point** as **Test Point M2** and place it at the left of coax cable block in the simulation model. The Test Point M1 is available by default at the left of load in the simulation model.

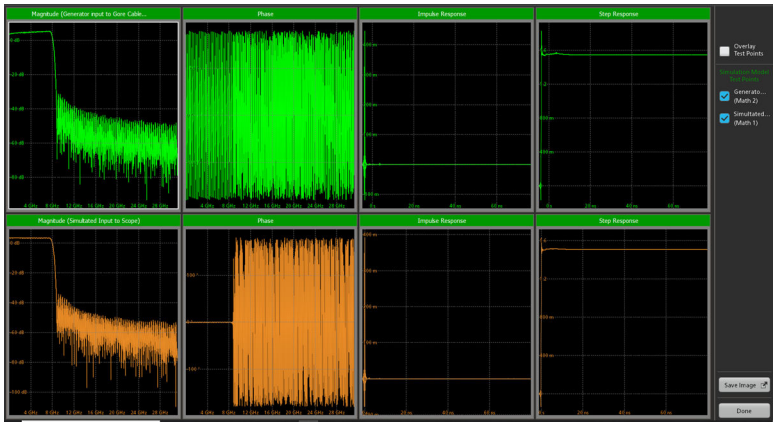
15. Double-tap the test point and set the **Global Delay** to **Keep**.



This maintains the exact time delay between the test point M2 and M1 waveforms.

16. Tap **Calculate SIM** to build the output math filters. These filters will be applied to the input source waveform in SIM to obtain the two test point waveforms in math 1 and math 2 waveform slots on the oscilloscope.

17. Tap **Plot Test Points**. Observe the frequency response, phase, impulse response, and step response of the two test point filters to validate if the simulation has reasonable looking filters.

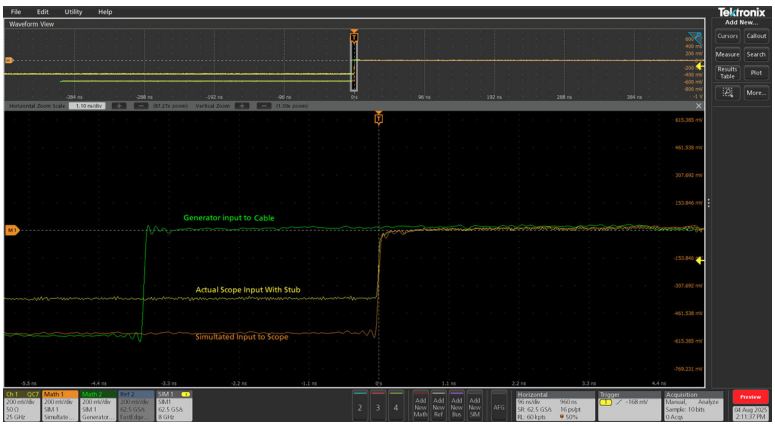


For example, if the filter impulse response is wild and not settled by the end of the filter length, then there may be a problem with the model, or there may be an issue such as trying to de-embed a deep null that is many dB down, resulting in high boost of many dB.

If there is major tilt prior to the edge of the step response, there may be an issue of poor internal linear extrapolation of the S-parameter blocks from their start frequency back down to DC.

In this case you may need to go back and actually measure the S-parameters down to DC, or perform your own extrapolation of the S-parameter to DC for both phase and magnitude and include that in the new S-parameter file to use in the SIM block.

Note that SIM does not support high pass model blocks for de-embedding, however, such high pass blocks are OK in the Simulation circuit.



Observe the test point simulated live waveforms on the oscilloscope display along with the original acquired waveform from Ch1 that was input to the oscilloscope. One of the test point filters is applied to Ch1 to get the simulated test point waveform in green at the input to the coax cable. The other filter was applied to Ch1 waveform, to get the orange simulated waveform at the output of the coax cable as with the stub de-embedded as if it is not present.

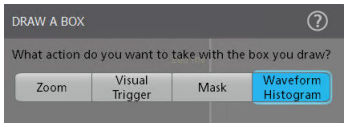
Waveform histogram

Histograms visually shows the distribution of how many sample points fall in selected section of waveform record. Waveform Histogram provides you tool to statistically analyze the waveforms using histograms in the graticule. This helps you to quickly determine how your device is responding to the inputs at discrete intervals of time and make the right decisions to change the device parameters.

Create a waveform histogram

Use the More button to create a waveform histogram on the instrument display.

1. Tap the **More** button and select the **Waveform Histogram** button in the **DRAW A BOX** menu window. This sets the DRAW A BOX function to Waveform Histogram mode.



2. Tap outside the menu to close the **DRAW A BOX** menu. The DRAW A BOX icon is highlighted to show waveform histogram draw mode is active.
3. Draw a rectangle on the waveform screen. The rectangle created is associated with the channel to which it was added (Stacked mode) or the active channel (Overlay mode).
4. Double-tap the **DRAW A BOX** button to add multiple histograms in the areas of interest of waveform record.
5. Continue drawing histograms on different waveforms:
   - In Stacked mode, you can draw histograms in different waveform slices.
   - In Overlay mode, select a channel before creating histograms for that channel.
6. When you are done drawing the histogram, tap anywhere to end the histogram draw function.
7. To draw a new histogram box again, tap the **DRAW A BOX** button again.

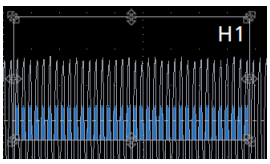
Edit histogram on the screen

Use the touchscreen or a mouse to directly edit a histogram, including moving, changing size, duplicating a histogram.

1. To move a histogram, drag the histogram box to a new position.

   NOTE: You cannot drag a waveform histogram from one waveform slice to another while in Stacked display mode.

2. To delete a histogram:
   - Touch and hold or right-click on the histogram and select **Delete Histogram** or **Delete All Histograms** from the menu.
   - Touch and hold or right-click on the histogram badge and select **Delete Histogram** or **Delete All Histograms** from the menu.
   - Select the histogram badge and press the Delete key on the keyboard.
3. To resize a histogram:
   a. Single-tap a histogram to enable resize histogram mode. Resize mode draws resize points on all four sides and the corners of a boundary area that contains the histogram.



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

NOTE: You cannot move a histogram while in resize mode.

Waveform Histogram guidelines

- The waveform histogram is based on a single source.
- You can draw unlimited histograms per source waveform.
- The waveform histogram are supported on analog channels, analog math waveforms, analog reference waveforms.
- The waveform histogram are not supported on Bus waveforms, Time-trend waveforms, and Math FFTs.
- A single waveform histogram can be either vertical or horizontal, but not both.
- Measurement Results Table includes result from histogram measurements that appear in the one or more histogram results badges.

  To add measurement results table for waveform histogram, see Add Results Table configuration menu (on page 184).

Waveform acquisition

Acquisition concepts

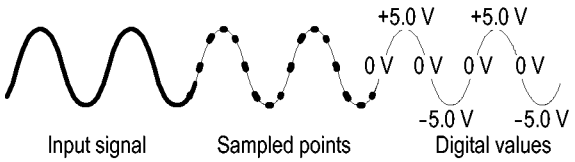
The Acquisition system sets which data points are used to acquire waveforms.

Acquisition hardware

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

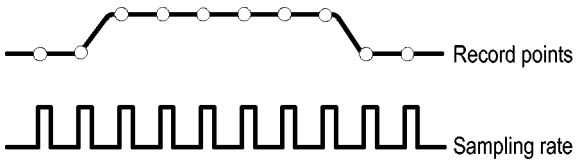
Sampling process

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



Real-Time sampling

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



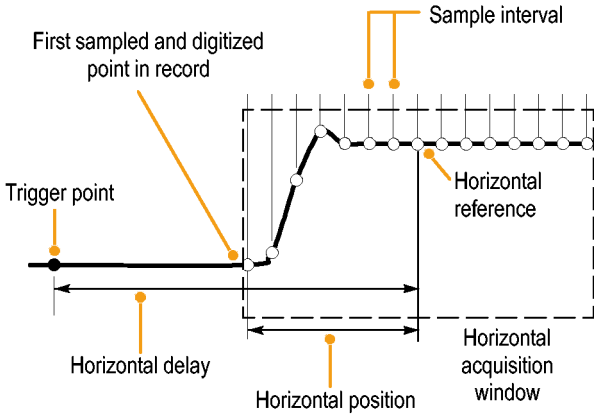
Interpolated Real-Time sampling

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

Waveform record

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

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



Interpolation

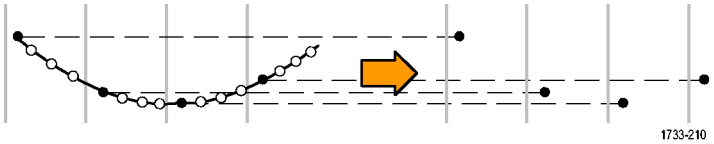
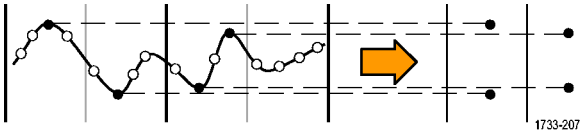
Your instrument can interpolate between the samples it acquires when it does not have all of the actual samples it needs to fill the waveform record. Linear interpolation computes record points between actual acquired samples by using a straight line fit.

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

Acquisition modes

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

How the acquisition modes work:

Acquisition mode	
Sample mode retains the first sampled point from each acquisition interval. Sample is the default mode. The instrument does no post processing of the acquired samples in this mode.	
Peak Detect mode 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.	
Envelope mode finds the highest and lowest record points over many acquisitions. Envelope uses Peak Detect for each individual acquisition.	

Acquisition mode	
Average mode calculates the average value for each record point over many acquisitions. Average uses Sample mode for each individual acquisition. Use Average mode to reduce random noise.	
FastAcq™ mode is helpful in finding elusive signal anomalies. Fast acquisition mode reduces the dead time between waveform acquisitions, enabling the capture and display of transient events such as glitches and runt pulses. Fast acquisition mode can also display waveform phenomena at an intensity that reflects their rate of occurrence.	
Fast Frame 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.	

Fast Frame concepts

Fast Frame 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 Fast Frame 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 Fast Frame 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

Fast Frame 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.

Fast Frame 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 Fast Frame on and selecting “Single Acquisition” to capture 100 frames, you use the readouts in the Fast Frame 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: To enable Fast Frame averaging, turn on Average Acquisition mode. When both Fast Frame and Average Acquisition mode are enabled, Fast Frame averaging is available. Summary Frame is not available in this setup.

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.

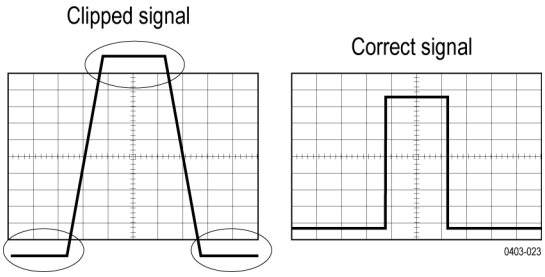
There are three options for interpolation:

- 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.

Scaling and positioning

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

Each waveform Slice or Graticule contains ten major divisions. This represents the maximum digitizing range of the instrument for any given vertical scale. Vertical waveform data that is outside (above and/or below) of the waveform Slice or Graticule is clipped; that is, the data values exceed the digitizing capability of the ADC at the current settings. This causes inaccuracies in amplitude-related measurements. For more information see Vertical acquisition considerations (on page 154).



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 155).

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

Vertical acquisition considerations

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

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

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

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

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

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

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

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

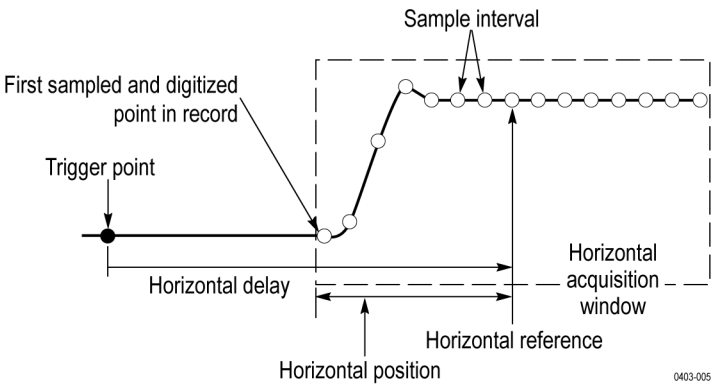
Horizontal acquisition considerations

The instrument lets you define the horizontal waveform display parameters that determine which segment of an incoming signal becomes the waveform record, following acquisition. For additional information, see "Waveform record" in Acquisition concepts (on page 151).

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.



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 (on page 318) to navigate to and select a location at which to save the file.
4. Tap **File Name** shows the name last used to save a file. The default name is Tek000. To change the file name, double-tap the file name and enter a new file name using the virtual keyboard.
5. Tap **Auto Increment File Name** to enable or disable automatic incrementing of a file name. Auto Increment File Name lets you save sequential files without needing to manually rename them each time. The count number is added to the end of the file name.
   - If Auto Increment File Name is enabled the **Count** defaults to 000 if there are no files at the specified location and file name that already use incremented file names. If there are files at the save location that already use the specified file name, and have already been saved using count increments, the Count field shows the next count value that will be added to the file name when the file is saved.
   - To change the starting count value, tap the Count field and use the assigned knob to change the value, or double-tap the field and use the virtual keypad to change the value.
6. Tap **Save As Type** and select the desired graphic image file type from the list.
7. Tap **Save** to save the screen image to the specified file name, location, and type.

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

Save a waveform to a file

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

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 (on page 318) 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 **Recall to Ref on Save** checkbox to recall the same reference waveform after saving it. This checkbox is unavailable when
   - Saving files as MATLAB (.mat) or SignalVu (.tiq).
   - When sources are analog channel and FastAcq is enabled.
   - Bus waveforms and when bus **Source** is set to **All** and one or more bus waveforms are present, only non-bus waveforms can be recalled as reference waveforms.
   - Autoref recall is not applicable to the waveforms saved by Act on Event.
7. Tap **Save As Type** and select the desired waveform data type from the list.
8. 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(s). Tap the source menu item to expand that item to show available waveform sources.
9. Tap **Save** to save the waveform to the specified file name, location, and type.

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

Save instrument settings to a file

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

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

The Save As configuration menu opens.

2. Tap **Setup** to open the Setup tab.
3. Tap **Save Location** or **Browse** to select the location to save the file.
   a. Tap the **Save Location** drop down arrow and select the location to save the file from a list of recent save locations. Or double-tap the field and use the virtual keyboard to enter a path to the save location.
   b. Tap **Browse** to open the Browse Save As Location configuration menu (on page 318) 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 **Save** 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 (on page 318) 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 **Save** 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 (on page 318) 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 **Save** button to immediately save the same type file again, without opening any menus.

Recall a reference waveform

Use this procedure to recall (load) and display a saved waveform as a Reference waveform. There is no set limit to the number of reference waveforms that you can load and display.

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

   The Recall configuration menu (on page 372) opens to the last location used to recall a waveform file. If the location is not available, the menu defaults to the instrument C drive.
2. Tap **Files of Type:** and select the file type from the list.

   The files list updates to show all files that match the selected file type.
3. Tap the file name in the list to add the file to the **File Name** field.
4. Tap **Open** to exit the menu and add the reference waveform to the screen.

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

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 (on page 313) menu.
2. Tap **Setup** to open the Setup tab.

   The Recall configuration menu opens to show the settings last used to recall a setup file.
3. Tap **Files of Type:** and select the file type from the list.

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

   As you navigate the folders, the files list area shows all files that match the file type selected in the Files of Type field.
5. Select the file to recall, using one of the following methods:
   - If the file was recently recalled, tap the drop down arrow in the **File Name:** field and select from a drop-down list of recently recalled files.
   - Select a file name in the main files list. The File Name field updates to show the selected file name.

   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 (on page 313) 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.
   - 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 or BER Eye Contour plot

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

NOTE: Mask can be recalled on BER Eye Contour plot by following similar steps as given here.

Prerequisites:

- Display an eye diagram plot, see Display an Eye Diagram plot (on page 97).
- An eye diagram mask file that defines at least the center segment and matches the time scale (unit interval) and vertical settings of the eye diagram.

Procedure:

1. Select eye diagram plot, on which mask to be recalled.
2. Tap **File** > **Recall** from the File menu.
3. Select type of file as **Mask** from the Recall menu.
4. Select the file you want to recall and tap **Recall**. Browse Mask File Location configuration menu. Or you can drag and drop the mask file into the eye diagram plot.

   If the file was recently recalled, tap the arrow in the **File Name** field and select a file from the list. Double-tap on a name in the File Name field to immediately recall the file and close the menu.

Supported mask file types

- **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.
- **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.
- **Tektronix Mask File Format (.xml)**: Tektronix' specific mask file format common across different mask testings.

When a selected mask file is not supported, a warning message will be displayed. To fix a mask file, edit the mask file format and add the string: `MASK:AUTOSET:USER:TYP Normalized`, if it is not present already.

Supported eye mask types

- **Absolute mask**: The points in the mask polygons are in the same coordinate units as the vector waveform or the eye diagram under analysis.
- **Relative mask**: The horizontal and vertical coordinates are normalized with respect to unit amplitude and unit interval of the vector waveform.

Menus and dialog boxes

All of the menus on your instrument and the available settings within each menu.

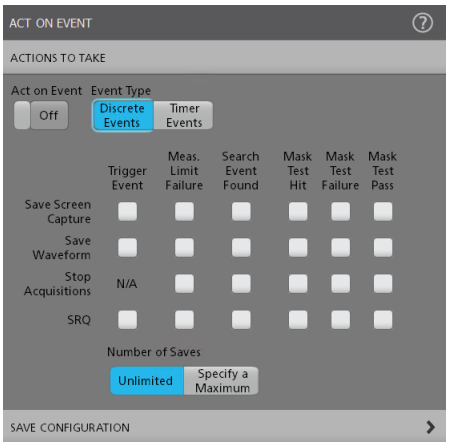
Act On Event configuration menu

Use this configuration menu to set the actions the instrument takes when specified conditions are met (trigger event, measurement limit failure, mask hit failure, and so on).

To open the **Act On Event** menu, select the **Act On Event** button in menus where this button is available.

Act On Event menu fields and controls

Displayed fields and controls can change depending on menu selections.



Field or control	Description
Actions To Take panel	
Act on Event On/Off	Allows you to configure Act On Event before enabling. If the maximum number of saves is hit, this control turns off.
Event Type	Select the event type as **Discrete Events** or **Timer Events** for saving screen capture, measurement result statistics logging, and saving waveforms. **NOTE:** The Timer events does not work with Single/sequence. It runs for Run/Stop sequencing only.
Timer Interval	Sets the time interval between two timer events. Available when **Event Type** is **Timer Events**.
Count	Sets number of timer events that should happen before stopping Run/Stop sequencing. Available when **Event Type** is **Timer Events**.
Save Screen Capture	Saves the screen image at the time of the event, to the format designated in the **Save Configuration** panel.

Field or control	Description
Save Waveform	Saves the waveform data at the time of the event, to the format designated in the **Save Configuration** panel.
Stop Acquisitions	Stops the instrument from acquiring any more data.
SRQ	This is a signal sent via the VISA connection to alert listeners that an event has occurred. Enabling this will send such an event when the action has occurred and if the status registers are configured correctly.
Number of Saves	Allows you to toggle between *Unlimited* or *Specify a Maximum* for number of saves. Default option selected is Unlimited.
Maximum	Enter the maximum number of saves in the text field. This is enabled when Number of Saves is set to Specify a Maximum.
Save Configuration panel	
Save Location	Choose the location of the saved files when an action triggers a save event.
Browse	Browse to the save location through a visual file tree.
File Name	Name the saved file. When a file is saved, the file name is followed by a time stamp.
Screen Capture Format	Choose format of the screen capture save files.
Waveform Format	Choose waveform format for the screen capture save files.
Source	Choose the source for the waveform save files.

Actions guidelines

- All actions are taken on a per acquisition basis. This means that if Save Screen Capture is checked in multiple columns and during a single acquisition the criteria is met in multiple columns, only a single screen capture is saved.
- For a timer event if there is no change in waveform dataset, the active measurement does not compute and log, instead shows an * symbol next to the measurement in the selected log file.
- The waveform save and screen capture save happens even if the waveform data is same for the two timer events.
- If there is no change in the waveform dataset and if any measurement configuration changed in between two timer events then the measurements are computed and the measurement result statistics are added in the log file (without * symbol).

Acquisition configuration menu

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

The Acquisition menu fields and controls

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

Displayed fields and controls can change depending on menu selections.

Field or control	Description
Run/Stop	Toggles the oscilloscope between constant acquisition (Run) and no acquisitions (Stop). When stopped, the oscilloscope shows waveforms from the last completed acquisition
Single/Seq	Acquires a single acquisition or a set number of acquisitions, then stops.
Clear	Erases acquired waveform data points from memory. Applies to all live acquisition waveforms.
Timebase Reference Source	Sets the time base source used by the oscilloscope to acquire and measure signals. Internal sets the oscilloscope to use an internal reference signal. External sets the oscilloscope to use a 10 MHz signal connected to the rear-panel Ref In connector. On some instruments External is divided into External (±2 ppm) and External (±1 k ppm) selections. The narrow External (±2 ppm) setting locks the timebase within 2 ppm of 10 MHz. The wide External (±1 k ppm) allows the timebase to be within 1 k ppm of 10 MHz.
Enable Acquisition History	Enables the acquisition history. By default, the option is unchecked. When checked, the History badge is enabled even if the acquisitions are running or stopped.
Fast Frame panel	Sets the Fast Frame acquisition mode. See Fast Frame Panel (on page 165).
Single Sequence/Stop After	Enables stopping acquisitions after a specified number of acquisitions. Only works when using the Single/Sequence button.
Acquisition Mode	**Sample** creates a record point by saving one or more samples during each acquisition interval. Sample mode is the default acquisition mode. The instrument does no post processing of the acquired samples in this mode.
	Peak Detect alternates between saving the highest sample in one acquisition interval and the lowest sample in the next acquisition interval. Useful for capturing fast and random events such as narrow waveform pulses.
	Envelope acquires and displays a waveform record that shows the extremes in variations over several acquisitions. The instrument saves the highest and lowest values in two adjacent intervals (similar to the Peak Detect mode). Unlike Peak Detect mode, the peaks are gathered over many trigger events.
	Average acquires and displays a waveform record that is the average result of several acquisitions. This mode reduces random noise.

Fast Frame Panel

The Fast Frame panel is accessed from the Acquisition badge menu.

To open the Acquisition menu:

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

Fast Frame menu fields and controls

Field or control	Description
Fast Frame	Enables or disables the Fast Frame mode. When enabled (On), the screen shows either the last acquisition frame captured, or will wait for acquisition completion of all the framed events.
Number of Frames	Shows the number of frames acquired. Tap on the field and use the multipurpose knob set the number of frames to acquire, or double-tap on the field and use the virtual keypad to enter a frame count. The maximum number of frames you can capture is displayed in the frame information readout area.
Frame information readout area	Displays frame acquisition information. The information changes depending on instrument settings, primarily the timebase, sampling rate and record length.
Summary Frame	Enables or disables creating and displaying a summary frame at the end of the frame record. When the current acquisition mode is set as Sample 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.

Fast Frame usage guidelines

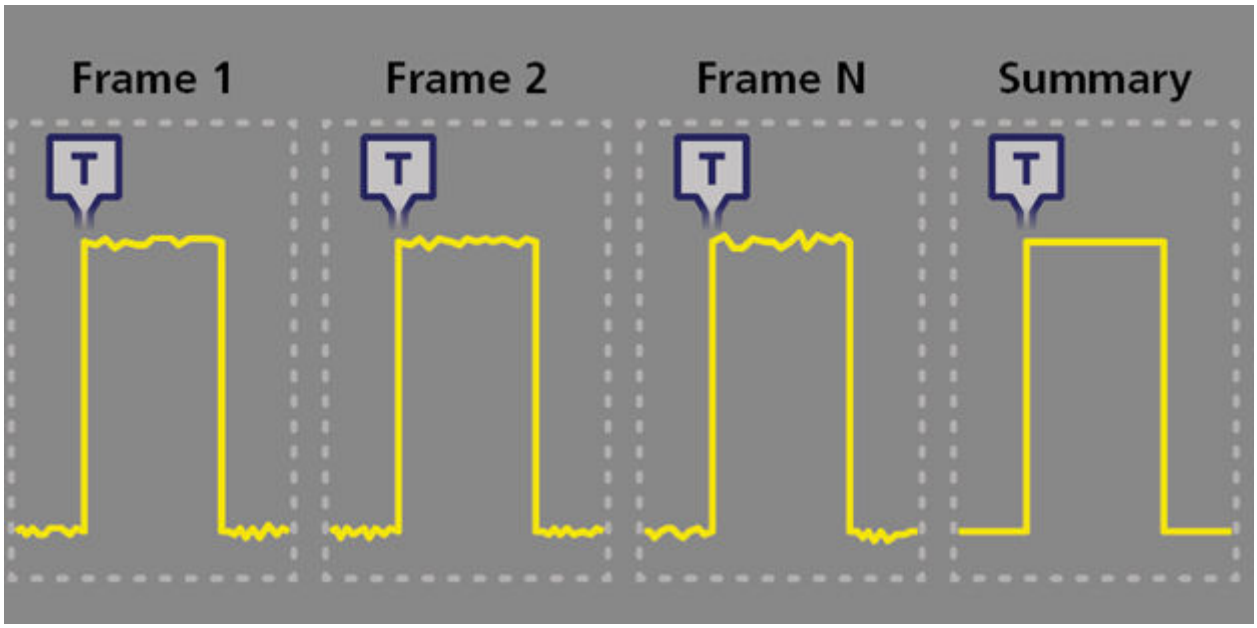
- Enabling Fast Frame mode disables FastAcq mode (if it was enabled). Likewise, enabling FastAcq mode disables Fast Frame mode (if it was enabled).
- You should define the trigger conditions to capture only the waveform, or waveform segment, in which you are interested, before enabling Fast Frame mode.
- You can use Fast Frame 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 Fast Frame 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 Fast Frame acquisitions.
- To obtain rapid statistical results on massive populations, open the **Acquisition** badge menu, select **Single Sequence**, and set the **Single Sequence/ Stops After** field to acquire multiple sets of frames and keep statistics on all of them. When the sequence finishes, the last set of frames is still in memory and the instrument has accrued statistical data for a large number of frames.
- Use the **Run/Stop** button to manually start and stop the acquisition. Frames are acquired continuously until stopped, at which point the last N frames are in memory.
- 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 Fast Frame waveforms as reference waveforms using the normal Save As process. Saved Fast Frame 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-Fast Frame) reference waveform, the Reference waveforms is always displayed regardless of the currently selected frame.
- When both Fast Frame and Average Acquisition mode are enabled, Fast Frame averaging is available. (Summary Frame is not available in this setup.)

Fast Frame Averaging

The Fast Frame acquisition function can be used to quickly acquire large sets of waveforms and create highly averaged waveform data. The kind of average waveform data that Fast Frame produces depends on how the Fast Frame acquisition is configured. This section will explain the different configurations.

Average Summary Frame

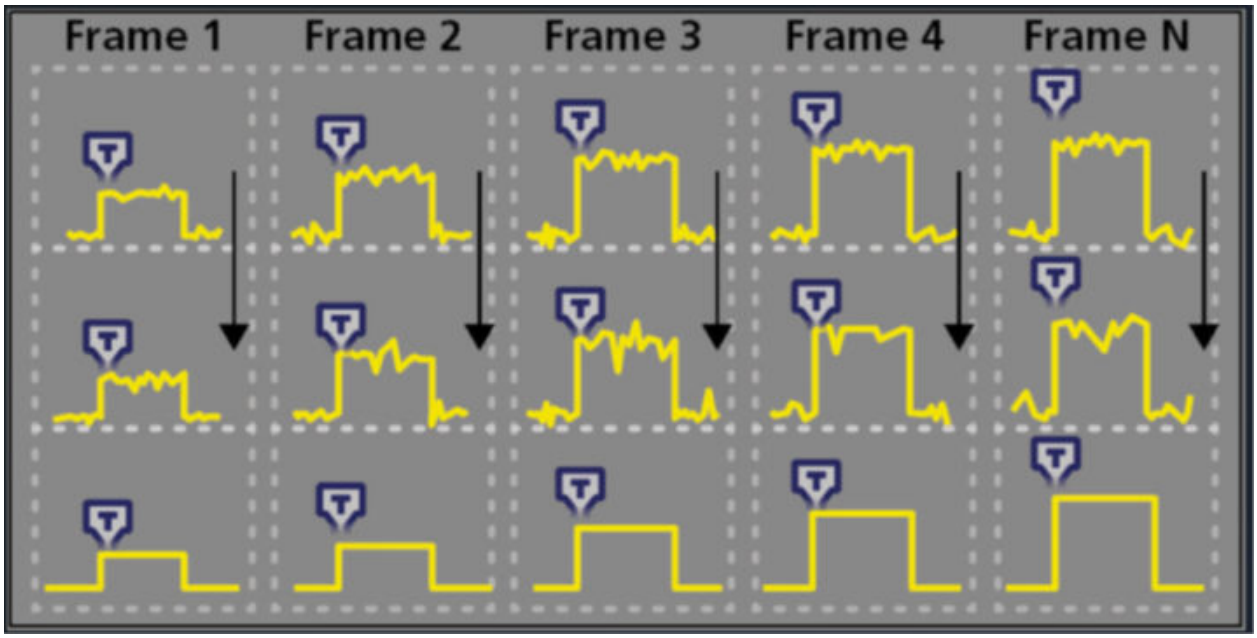
When the Acquisition Mode is set to Sample, Fast Frame is enabled, and the Summary Frame is enabled, the Summary Frame will contain a waveform that is the average across all frames in the acquired Frame Set. In this configuration frames will be averaged together horizontally from Frame 1 through Frame N to produce a Summary Frame at the end of the Frame Set, as illustrated in the following image.



The Average Summary Frame is useful for quickly acquiring many waveforms and producing an average waveform. It is especially useful when the trigger events happen very rapidly and would normally be missed without the use of Fast Frame.

Orthogonal Fast Frame Averaging

When the Acquisition Mode is set to Average and Fast Frame is enabled, averaging will be performed between frame sets rather than between frames contained in the same set. This is known as Orthogonal Fast Frame Averaging. In this mode, frame averaging is performed vertically with Frame 1 of the Frame Set being averaged with Frame 1 of the next Frame set, Frame 2 with Frame 2 continuing all the way to Frame N as seen in the following image.



As each Frame Set is acquired, the acquisition count that makes up the average waveform contained within each frame increases by one. This results in a final Frame Set where each frame contains averaged waveform data. This differs from the Average Summary Frame, where only the Summary Frame contains average waveform data.

Orthogonal Fast Frame Averaging is useful for acquiring repeatable, multi-step processes where averaging is required to get a higher signal to noise ratio.

Orthogonal Fast Frame Averaging guidelines

- When Orthogonal Fast Frame Averaging is enabled, Summary Frame is not available.
- When the oscilloscope is operating in Continuous Run mode, if the acquisition is stopped before the current Frame Set has completed acquiring, the oscilloscope will attempt to complete the Frame Set before stopping. If no more trigger events occur and the Frame Set cannot be completed, the acquisition will time out and the acquisition will stop without completing the Frame Set.

Fast Frame badge

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

The Fast Frame badge is opened when you enable the Fast Frame mode in the Fast Frame panel (on page 165) of the Acquisition configuration menu.

Fast Frame badge fields and controls

Field or control	Description
Frame count	Shows the total number of frames captured at the top of the badge.

Field or control	Description
Selected Frame	Shows the selected frame in all displayed waveforms (analog, digital, and math). Use Multipurpose A knob to scroll through and select specific frames. To enter a specific frame number to display, use the Selected Frame field in the Fast Frame configuration menu (on page 169). Frame scrolling uses a wrap-around method. For example, when you reach the end of the frame list, scrolling further displays frame 1. If you have enabled overlay mode, the current frame's waveform is highlighted in blue in all displayed waveforms (analog, digital, and math).
Reference Frame	Selects a frame in the current frame acquisition that you want to use as a time reference against which to compare the selected frame. Displays a Delta readout that shows the difference between the Selected Frame and the Reference Frame. To directly enter a specific frame number to display, use the **Reference Frame** field in the Fast Frame configuration menu (on page 169). Available only when **Include Reference Frame in Badge** is selected in the Fast Frame configuration menu.
Time stamps	Shows the time difference between frame one and the Selected frame. If **Include Reference Frame in Badge** is enabled, each frame readout area (Selected and Reference) in the badge shows the time difference between frame one and the Selected and Reference frames. When both Selected and Reference frames are enabled, a **Delta** time readout shows the time difference between the Selected frame and the Reference frame. If the difference time displayed goes beyond 10 seconds, then digits are dropped from the right as needed and the displayed timestamp is rounded. Note that this is just for UI display purposes. The full, precise value is stored and available.
Navigation buttons	Tap to display the previous or next frame. Navigation buttons are only displayed when acquisition is set to Stopped.
Summary	Displays the summary frame. the Summary frame button is only displayed when the Summary Frame button is enabled in the Acquisition menu Fast Frame panel.

Fast Frame badge configuration menu

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

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

Fast Frame configuration menu fields and controls

Field or control	Description
Frame count	Shows the total number of frames captured at the top of the badge.

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

Add Measurements configuration menu overview

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

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

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

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

To change individual measurement settings, double-tap the Measurement badge to open a configuration menu for that measurement. See Measurement configuration menu overview (on page 300).

Add Measurements menu fields and controls

Field or control	Description
Measurement tabs	The tabs along the top organize measurements by their type. The Standard tab is the default set of measurements that are built in to the instrument. Other tabs are shown when you install measurement options. See The Jitter tab (Advanced Jitter and Eye Analysis) (on page 173).
Measurement description (graphic and text)	Shows a graphic and short description of the selected measurement. Use this information to verify that the selected measurement is correct for what you want to measure.
Source	Selects the measurement source. If the measurement requires more than one source (for example, Skew, Phase), the menu shows two source fields from which to select.
Add button	Adds the selected measurement as a measurement badge to the Results bar. You can also double-tap a listed measurement to add it to the Results bar.

Standard tab measurement panels

Panel	Description
Amplitude Measurements panel	Tap the panel bar to list the available amplitude measurements. Touch and drag the list to scroll through all measurements. See Amplitude Measurements panel (on page 171).
Time Measurements panel	Tap the panel bar to list the available time measurements. Touch and drag the list to scroll through all measurements. See Timing Time Measurements panel (on page 180).
Jitter Measurements panel	Tap the panel bar to list the available standard jitter measurements. See Jitter Measurements panel (on page 173). **NOTE:** If you have installed the Advanced Jitter and Eye Analysis option, the **Jitter Measurements** panel is removed from the **Standard** measurement tab and is replaced with a **Jitter** tab at the top of the **Add Measurements** menu. See The Jitter tab (Advanced Jitter and Eye Analysis) (on page 173).

Other tab measurements (require optional licenses)

Add Measurement tab	Description
Jitter	Advanced Jitter and Eye Analysis measurements option. Provides triggers and measurements for advanced jitter and eye analysis. See The Jitter tab (Advanced Jitter and Eye Analysis) (on page 173).

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. The measurement annotations are supported for all amplitude measurements.

To open the Amplitude Measurements panel:

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

To add a measurement to the Results bar:

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

The Amplitude Measurements panel measurements

Measurement	Description
AC RMS	The true Root Mean Square voltage, minus any DC component, of the waveform data points that are above the Mean signal level. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Amplitude	The difference between the Top value and the Base value. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Area	The area between the waveform and the Mean value, calculated by integrating the data points. The area above ground (0 V) is positive, while the area below ground is negative. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Base	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. **NOTE:** Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated
Maximum	The maximum data point value. You can take this measurement on each cycle in the waveform record or across the entire waveform record.
Mean	The arithmetic mean of all data points over the measurement region. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Minimum	The minimum data point value. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Negative Overshoot	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. **NOTE:** Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated
Peak-To-Peak	The absolute difference between the Maximum and Minimum amplitudes in the measurement region. You can take this measurement on each cycle in the waveform record or on the entire waveform record.
Positive Overshoot.	The difference between the Maximum value and the Top value, divided by the Amplitude, and multiplied by 100 to express the measurement as a percentage of amplitude. You can take this measurement on each cycle in the waveform record or on the entire waveform record. **NOTE:** Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated
RMS	The true Root Mean Square (The square root of the mean (average) value of the squared function of the waveform data points.) You can take this measurement on each cycle in the waveform record or on the entire waveform record.

Measurement	Description
Top	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. **NOTE:** Changing your Base Top Method in the Reference Levels panel of the Measurement configuration menu may change how this value is calculated

See also

Measurement configuration menu overview (on page 300)

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) (on page 173).

Jitter Measurements panel measurements

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

See also

Measurement configuration menu overview (on page 300)

Jitter tab (Advanced Jitter and Eye Analysis)

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 the Advanced Jitter and Eye Analysis option.

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

Measurement	Description
DCD	Duty cycle distortion. The peak-to-peak amplitude for the component of the deterministic jitter correlated with the signal polarity. The measurement is taken on the entire record.
DDJ	Data dependent jitter. The peak-to-peak amplitude for the component of the deterministic jitter correlated with the date pattern in the waveform. The measurement is taken on the entire record.
ISI	Inter Symbol Interference. The ISI value is defined as the range (difference between the maximum and minimum value) of DDJ-per-edge values appearing across all rising edges, or the range of DDJ-per-edge values across all falling edges, whichever is greater.
DJ	Deterministic jitter. The peak-to-peak amplitude of all timing errors that exhibit deterministic behavior. The measurement is taken across the entire record.
DJ-δδ	Dual-Dirac deterministic jitter. The deterministic jitter on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal-magnitude Dirac functions. The measurement is taken on the entire record.
SJ@Freq	SJ@Freq refers to sinusoidal jitter occurring at or near a frequency. It is defined as the peak-to-peak amplitude of the sinusoidal components of deterministic jitter within a designated frequency band around the target frequency.
F/2	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (date rate) divided by two. The measurement is taken on the entire record.

Measurement	Description
F/4	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (date rate) divided by four. The measurement is taken across the entire record.
F/8	The peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (date rate) divided by eight. The measurement is taken on the entire record.
J2	The total jitter at a bit error rate of 2.5e-3 (TJ@2.5e-3). The measurement is taken on the entire record.
J9	The total jitter at a bit error rate of 2.5e-10 (TJ@2.5e-10). The measurement is taken on the entire record.
CC-Jitter	Cycle to Cycle Jitter measures the clock period change between any two adjacent cycles. The measurement is taken on the entire record.
Jitter Summary	Adds multiple jitter measurements to the measurement badge, and displays Bathtub, TIE Spectrum trend plot, Composite Jitter Histogram, Eye Diagram plots on the screen.
NPJ	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.
Phase Noise	The RMS magnitude of all integrated jitter falling within a specified offset range of the fundamental clock frequency. The measurement is taken on the entire record.
PJ	Periodic jitter. The peak-to-peak amplitude of the sinusoidal components of deterministic jitter that are uncorrelated to the data pattern. The measurement is taken on the entire record.
RJ	Random jitter. The RMS magnitude of all random timing errors following a Gaussian distribution. The measurement is taken on the entire record.
RJ-δδ	Dual-Dirac random jitter. The random jitter on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal-magnitude Dirac functions. The measurement is taken on the entire record.

Measurement	Description
RJ(h)	RJ(h) is the portion of random jitter (RJ) attributable to random horizontal displacement of the localized waveform. Since RJ(h) and RJ(v) are uncorrelated, the total random jitter is calculated as: $RJ = sqrt\ (RJ(h)^2 + RJ(v)^2)$ This measurement is performed across the entire record length.
RJ(v)	RJ(v) is the portion of random jitter (RJ) caused by random vertical (amplitude) noise. When the waveform has a finite slew rate at its transitions, amplitude noise is converted into timing variation, which appears as jitter. Because RJ(h) and RJ(v) are uncorrelated, the total random jitter is calculated as: $RJ = sqrt\ (RJ(h)^2 + RJ(v)^2)$ This measurement is performed across the entire record length.
PJ(v)	PJ(v) is the portion of periodic jitter (PJ) attributable to periodic vertical noise. When the waveform has a finite slew rate at its transitions, amplitude noise is converted into timing variation, which appears as jitter. For any given frequency, PJ(h) and PJ(v) are correlated and are added algebraically. This measurement is performed across the entire record length.
PJ(h)	PJ(h) is the portion of periodic jitter (PJ) attributable to periodic horizontal displacement of the waveform. For any given frequency, PJ(h) and PJ(v) are correlated and are added algebraically. This measurement is performed across the entire record length.
SRJ	Sub-rate jitter. The composite jitter due to periodic components at ½, 1/4, and 1/8 of the data rate. The measurement is taken on the entire record.
TIE	Time Interval Error. The difference in time between an edge in the source waveform and the corresponding edge in a recovered reference clock signal. The measurement is taken on each waveform edge.
TJ@BER	Total jitter at a specified bit error rate. The predicted 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.

NOISE MEASUREMENTS

Measurement	Description
RN	Random Noise (RN) is the RMS magnitude of all non-deterministic, Gaussian-distributed vertical deviations from the nominal bit amplitude at the specified unit interval (UI) offset of each bit interval.

Measurement	Description
TN@BER	Total Noise at BER (TN@BER) is the total noise at a specified bit error rate (BER). This extrapolated value statistically predicts the peak-to-peak vertical eye closure, at the specified horizontal bit offset, that will be exceeded only with a probability equal to the BER. It is typically not equal to the actual vertical eye closure observed during a given acquisition interval.
NPN	Non-Periodic Noise (NPN) is the dual-Dirac magnitude of the random component of bounded uncorrelated noise (BUN). BUN excludes deterministic noise (DN), data-dependent noise (DDN), and random noise (RN). This measurement is performed across the entire record.
DN	Deterministic Noise (DN) is the peak-to-peak amplitude of all vertical deviations from the nominal bit amplitude that exhibit deterministic (nonrandom) behavior. This measurement is performed across the entire record.
Noise Summary	Noise Summary is a group of measurements that includes: ▪ TN@BER ▪ RN ▪ DN ▪ PN ▪ DDN ▪ DDN(0) ▪ DDN(1) ▪ Height@BER ▪ Unit Amplitude
DDN(1)	Data-Dependent Noise for 1 Bits (DDN(1)) is the peak-to-peak amplitude of all bit-pattern-correlated vertical variations of logic-high bits relative to the nominal high level at the center of the eye. A single DDN(1) value is calculated for each acquisition.
DDN(0)	Data-Dependent Noise for 0 Bits (DDN(0)) is the peak-to-peak amplitude of all bit-pattern-correlated vertical variations of logic-low bits relative to the nominal low level at the center of the eye. A single DDN(0) value is calculated for each acquisition.
DDN	Data-Dependent Noise (DDN) is the total vertical eye closure caused by bit-pattern-correlated vertical variations at the center of the eye. It is calculated as the sum of the positive peak DDN(0), relative to the nominal low level, and the negative peak DDN(1), relative to the nominal high level. A single DDN value is calculated for each acquisition.
RN(v)	RN(v) is the portion of random noise (RN) attributable to random vertical displacement of the waveform. Since RN(v) and RN(h) are uncorrelated, the total random noise is calculated as: $RN = \mathrm{sqrt}\ (RN(v)^2 + RN(h)^2)$ This measurement is performed across the entire record length.

Measurement	Description
RN(h)	RN(H) is the portion of RN attributable to random jitter. Since the signal has a nonzero slew rate at the measurement point, this timing variation appears as an equivalent vertical voltage displacement. Because RN(V) and RN(H) are uncorrelated, $RN = sqrt\ (RN(h)^2 + RN(v)^2)$ This measurement is performed across the entire record length.
Unit Amplitude	Unit Amplitude is the nominal eye amplitude measured at the horizontal center of the eye. The nominal high level is the mean value of the distribution represented by DDN(1), and the nominal low level is the mean value of the distribution represented by DDN(0). Unit Amplitude is the difference between the nominal high and nominal low levels. This measurement is performed across the entire record length.
PN	Periodic Noise (PN) is the peak-to-peak amplitude of the portion of deterministic noise that is periodic but whose period is not correlated with any data pattern in the waveform. This measurement is performed across the entire record.
PN(h)	PN(h) is the portion of periodic noise (PN) attributable to periodic jitter. Because the signal has a nonzero slew rate at the measurement point, this timing variation appears as an equivalent vertical voltage displacement. For any given frequency, PN(v) and PN(h) are correlated and are added algebraically. This measurement is performed across the entire record length.
PN(v)	PN(v) is the portion of periodic noise (PN) attributable to periodic vertical displacement of the waveform. For any given frequency, PN(v) and PN(h) are correlated and are added algebraically. This measurement is performed across the entire record length.

Eye Measurements panel

Measurement	Description
Eye Height	The minimum vertical eye opening at the center of the recovered unit interval. The measurement is taken on the entire waveform record.
Eye High	The amplitude of a high (one) bit measured at a specified location within the recovered unit interval. The measurement is taken on each high bit in the waveform record.
Eye Low	The amplitude of a low (zero) bit measured at a specified location within the recovered unit interval. The measurement is taken on each low bit in the waveform record.
Eye Width	The minimum horizontal eye opening at the center of the recovered unit interval. The measurement is taken on the entire waveform record.

Measurement	Description
EyeHeight@BER	The predicted vertical eye opening that will be violated with a probability equal to the bit error rate. The measurement is taken on the entire waveform record.
EyeWidth@BER	The predicted horizontal eye opening that will be violated with a probability equal to the bit error rate. The measurement is taken on the entire waveform record.
Q-Factor	The ratio of the vertical eye opening to RMS vertical noise measured at a specified location in the recovered unit interval. The measurement is taken on the entire waveform record.

Amplitude Measurements panel

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

Timing Measurements panel

Measurement	Description
Data Rate	Data Rate is the reciprocal of Unit Interval. This measurement is the same as Data Rate measurement under Standard category time panel.
Pattern Length	Number of symbols that constitute one full repetition of a sequence within a signal.
SSC Freq Dev	The spread spectrum clock frequency deviation. This measurement enables a time trend plot of the spread spectrum clock modulation profile. The measurement is taken on each cycle of the entire record.
SSC Modulation Rate	The modulation frequency of a spread spectrum clock. The measurement is taken on each cycle of the entire record.
SSC Slew Rate	The rate of change of the modulating 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 300)

Standard measurements tab

The **Standard** tab lists the default set of amplitude,Time, 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 171)

   Timing Time Measurements panel (on page 180)

   Jitter Measurements panel (on page 173)
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) (on page 173).

Time Measurements panel

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

To open the Time Measurements panel:

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

To add a measurement to the Results bar:

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

Time Measurements panel

Measurement	Description
Burst Width	The duration of a series of adjacent crossings of the Mid reference level. Bursts are separated by a specified idle time. The measurement is taken on each burst in a waveform record. **NOTE:** This measurement can also be taken on digital signals.
Data Rate	Data Rate is the reciprocal of Unit Interval. The measurement is taken on each bit of the waveform record. **NOTE:** This measurement can also be taken on digital signals.
Delay	The time difference between the mid reference (default 50%) amplitude point of two different waveforms. You can specify the signal edges to measure in the measurement's Configuration menu. The measurement requires two sources.
Duration N-Periods	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. **NOTE:** This measurement can also be taken on digital signals.
Fall Time	The time required for the trailing edge of the first pulse in the measurement region to fall from the high reference value (default = 90%) to the low reference value (default = 10%). The measurement is taken on each cycle of the waveform record.
Falling Slew Rate	The rate of change (in volts/second) as an edge transitions from a high reference level to a low reference level. The measurement is taken on each cycle of the record in the measurement region.
Frequency	The frequency of the waveform. Frequency is the reciprocal of Period (Frequency = 1/Period). **NOTE:** This measurement can also be taken on digital signals.
High Time	The time the signal remains above the Top reference level. The measurement is taken on each cycle in the waveform record.

Measurement	Description
Hold Time	The time between the specified Mid reference level crossing on the clock signal to the closest Mid reference level crossing on the specified data signal. The measurement is taken on each specified clock edge in the waveform record. **NOTE:** This measurement can also be taken on digital signals.
Low Time	The time the signal remains below the Base reference level. The measurement is taken on each cycle in the waveform record.
Negative Duty Cycle	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. **NOTE:** This measurement can also be taken on digital signals.
Negative Pulse Width	The distance (time) between the mid reference (default 50%) amplitude points of a negative pulse. The measurement is taken on each cycle in the waveform record or measurement region. **NOTE:** This measurement can also be taken on digital signals.
Period	The time between two adjacent crossings of the Mid reference level (one cycle) of the waveform. The measurement is taken on each cycle of the waveform record or measurement region. **NOTE:** This measurement can also be taken on digital signals.
Phase	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. **NOTE:** This measurement can also be taken on digital signals.
Positive Duty Cycle	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. **NOTE:** This measurement can also be taken on digital signals.
Positive Pulse Width	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. **NOTE:** This measurement can also be taken on digital signals.
Rise Time	The time required for the leading edge of the first pulse in the measurement region to rise from the low reference value (default = 10%) to the high reference value (default = 90%). The measurement is taken on each cycle of the waveform record.

Measurement	Description
Rising Slew Rate	The rate of change (in volts/second) as an edge transitions from a low reference level to a high reference level. The measurement is taken on each cycle of the record in the measurement region.
Setup Time	The time between the specified Mid reference level crossing on the data signal to the closest Mid reference level crossing on the specified clock signal. The measurement is made on each specified clock edge in the waveform record.
Skew	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. **NOTE:** This measurement can also be taken on digital signals.
Time Outside Level	The time the specified signal remains above the Top reference level and/or below the Base reference level. The measurement is made on each occurrence in the waveform record.
Time to Min	Time to Min is the amount of time from the trigger point to the minimum data point. This measurement can be made across the entire record or on each cycle in the record.
Time to Max	Time to Max is the amount of time from the trigger point to the maximum data point. This measurement can be made across the entire record or on each cycle in the record.
Unit Interval	The time difference between two successive bits. The measurement is taken on each bit of the waveform record. **NOTE:** This measurement can also be taken on digital signals.
TDR	Time Domain Reflectometry (TDR) is the arithmetic mean of all the data points in the impedance waveform which is computed from the voltage waveform over the measurement region. The measurement also determines the capacitance and inductance over the same measurement region. If the signal type is set to Differential, the measurement requires two sources.

See also

Measurement configuration menu overview (on page 300)

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
Measurements	Display a table of all the measurements in the Results bar.
Search	Display a table of all defined searches with each search shown on its own tab.
Bus Decode	Display a table of the bus decode results.
Add	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 Results Table configuration menu (on page 189)

Search Results table menu (on page 192)

Bus Decode Results table configuration menu

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

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

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

Bus Decode Results table menu

Field or control	Description
Column Visibility panel	
Check boxes for columns in table to display	Select or clear individual check boxes to add or remove that column from the table. The listed columns depend on the selected bus table in the Bus Decode Results table. Select the tab of the bus, at the top of the table, to which you want to make changes, then double-tap the table to open the configuration menu for that table. Available check boxes are ▪ Start Time ▪ Parity (h) ▪ Sync (h) ▪ Unframed (h) ▪ Packet Header (h) ▪ Error ▪ Packet Data (h) ▪ Stop Time ▪ Packet Trailer (h)
Other panel	
Save Table	Opens the Save As dialog to let you save the table data to a comma-separated values (.csv) file. See Save As configuration menu (Bus Decode Results table) (on page 187).

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.
- When both Fast Frame and one or more buses are in use, the Bus Decode Results table will display the decoded data for the selected frame. You can switch through frames and see the decoded data by using the previous/next buttons in the Fast Frame badge. The table does not show the decoded data for all the frames at the same time.

NOTE: To view decoded data for all the frames: After acquiring multiple frames of data and decoding, right click on the bus decode results table and click on **Save As** to export the decoded data of all the frames into a .csv file (you can view the .csv file for offline data analysis). You can access the decoded packet data of all the frames at the same time in the exported file. In the exported file, the frame number column will uniquely identify the frames with the decoded data packets.

- After the decoded packets are displayed in the table, select any row and the waveform zoom points to the respective packet in the bus display. Vice versa, when you move the zoom window, the corresponding packet in the table will be selected.

Save As configuration menu (Bus Decode Results table)

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

To open the Save As configuration menu:

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

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

Field or control	Description
Save Location	Sets the location to which to save the file. The default value is the last location to which a file of this type was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, for the current save type, up to a maximum of 20 locations.
Browse	Tap to open the Browse Save As Location configuration menu, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu (on page 318).
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a default name created by the instrument if this file type has not previously been saved with a custom file name. The default file name uses the format Tek000. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save as Type	You can only save table results as comma separated values (.csv) files.

Annotation and navigation on waveform plots/data and results table

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 ΔV/ ΔI based on the measurement configuration, where V/I is static and ΔV/ ΔI 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.

Frequency Response Analysis (FRA) Results table configuration menu

Custom Results Table configuration menu

Field or control	Description
Table Settings panel	
Column Visibility	Select or clear individual check boxes to add or remove that column from the table.
Save panel	
Save Table	Opens a menu to save the Custom Results Table data to a comma-separated value (CSV) file. See Save As configuration menu, Custom Results Table (on page 194).

Navigation on Bar Graph and Harmonics Results Table

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.

Measurement Results Table configuration menu

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

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

Measurement Results Table menu

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

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
Save Location	Sets the location to which to save the file. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, for the current save type, up to a maximum of 20 locations.
Browse	Tap to open the Browse Save As Location dialog, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu (on page 318).
File Name	The file name assigned to the file. The default file name is Tek000. The file name increments on the next save (Tek001, Tek002, and so on). Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the on-screen keyboard and enter a file name.
Save as Type	You can save only table results as comma separated values (.csv) files.

Peak Marker results table configuration menu

Use this menu to configure the content of the Peak Marker results table.

To open the Peak Marker results table configuration menu, double-tap anywhere in the table. If there are multiple Peak Marker results tables, tap the tab of the table to configure and then double-tap anywhere in that table.

Peak Marker results table fields and control

Field or control	Description
Frequency Resolution	Toggles the frequency resolution **Auto** or **Precise**. The default is **Auto**. **Auto**: Shows the frequency with the same precision as shown in the Spectrum View display. **Precise**: Shows the frequency down to single Hz resolution.
Save Table	Opens the Save As dialog to let you save the table data to a comma-separated values (.csv) file. See Save As configuration menu (Peak Marker results table) (on page 192).

Right click menu for Peak Marker results table

Field or control	Description
Configure Table	Configure Peak Marker table.
Save Table	Opens the Save As dialog to let you save the table data to a comma-separated values (.csv) file. See Save As configuration menu (Peak Marker results table) (on page 192).
Delete Table	Tap to delete the table.

Save As configuration menu (Peak Marker results table)

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

To open the Save As configuration menu:

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

Save As configuration menu (Peak Marker results table)

Field or control	Description
Save Location	Sets the location to save the file. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter the location. Use an external keyboard or double-tap on the file name to open the on-screen keyboard and enter the path details. You can also use the Browse button to open a menu to navigate to and select the location to which to save the file. Tap the down arrow icon on the right end of the field to open the list of recently saved file locations, for the current save type, up to a maximum of 20 locations.
Browse	Tap to open the Browse Save As Location dialog, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu (on page 318).
File Name	The file name assigned to the file. The default file name is Tek000. The file name increments on the next save (Tek001, Tek002, and so on). Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the on-screen keyboard and enter a file name.
Save as Type	You can only save a Peak Marker results table to a comma-separated value (.csv) format file.

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
Table Settings	
Location Timestamp Resolution	Sets the search mark timestamp resolution to display. The timestamp shows the time of each mark relative to the first acquired search mark. **Short** shows a rounded version of the timestamp data. **NOTE:** The actual timestamp data is not truncated by the Short setting, and is retained in the acquisition data. **Precise** displays the full timestamp value by individual time units (seconds, milliseconds, microseconds, and so on).
Delta resolution	Sets the timestamp resolution to display for mark deltas. Delta shows the time difference between each search mark. **Short** shows the delta time between marks as a rounded version of the timestamp data. **NOTE:** The actual timestamp data is not truncated by the Short setting, and is retained in the acquisition data. **Precise** displays the full delta timestamp value by individual time units (seconds, milliseconds, microseconds, and so on).
Save Table	Opens a menu with which to save the results table data to a file. See Save As configuration menu (Measurement Results Table) (on page 190).
Column Visibility panel	
Check boxes to select columns to display	Select or clear individual check boxes to add or remove that column from the table.

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.
- After the search events are displayed in the table, select any row and the waveform zoom points to the respective event in the display. Vice versa, when you move the zoom window, the corresponding row will be selected in the table.

Save As configuration menu, Custom Results Table

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

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

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

Save As configuration menu, Custom Results Table

Field or control	Description
Save Location	Sets the location to save the file. The default value is the previous save location. Tap on the file path and enter the location. Use an external keyboard or double-tap on the file name to open the virtual keyboard and enter the path details. You can also use the **Browse** button to open a menu to navigate to and select the save location. Tap the down arrow icon on the right end of the field to open the list of recently saved file locations for the current save type. Up to 20 recently saved file locations are stored.
Browse	Tap to open the Browse Save As Location dialog, to navigate to and select the location to save the file. See Browse Save As Location configuration menu (on page 318).
File Name	Enter the file name. The default file name is Tek000. The file name increments on the next save (Tek001, Tek002, and so on). Double-tap on the file name to open the virtual keyboard or use an external keyboard to enter the name.
Save as Type	You can only save a Custom Results Table to a comma-separated value (CSV) file.

AFG configuration menu

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

To open the AFG configuration menu:

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

Arbitrary/Function Generator overview

The function generator provides output of predefined waveforms up to 100 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, a network drive or TekDrive).

Arbitrary/Function Generator menu fields and controls

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

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

Field or control	Description
Output	Tap to toggle between the output **Off**, **Continuous**, or **Burst**.
Cycles	Specify the number of cycles in the text field. Default cycle value is **1**. Available when **Output** is **Burst** When Output is selected as Burst, a Burst button is displayed on the AFG badge. Click the Burst button on the badge to generate the signal burst.
Waveform Type	Tap to select an available waveform from the list.
Load	Tap to open the Directory configuration menu. Navigate to and select a waveform file to load into the AFG memory. You can navigate to TekDrive to load saved waveform files. Available when **Waveform Type** is **Arbitrary**. **NOTE:** Differential output not available with DC level waveform type.
Waveform File	Shows the loaded waveform file path and name. Tap to select a waveform file to load into the AFG waveform memory from the drop-down list of the last 20 waveforms that have been loaded using the **Load** button.
Symmetry	Sets the symmetry of the ramp using the keypad or multipurpose knob. Available when the **Waveform Type** is **Ramp**.
Width	Sets the width of the pulse using the keypad or multipurpose knob. Available when the **Waveform Type** is **Pulse**.
Duty Cycle	Sets the duty cycle of the square wave using the keypad or multipurpose knob. Available when the **Waveform Type** is **Square**.
Frequency	Sets the frequency of the waveform using the keypad or multipurpose knob. The frequency range is 0.1 Hz to 100 MHz (only for sine wave) and for other waveforms the range is 50 MHz, in increments of 0.1 Hz.
Period	Sets the period of the waveform using the keypad or multipurpose knob.
Amplitude	Sets the amplitude of the waveform using the keypad or multipurpose knob
Offset	ets the offset of the waveform using the keypad or multipurpose knob.
High Level	Sets the High signal amplitude of the waveform using the keypad or multipurpose knob
Low Level	Sets the low signal amplitude of the waveform using the keypad or multipurpose knob.
Load Impedance	Tap to select either **50 Ω** or **High Z** (1 MΩ) output load impedance.

Field or control	Description
Add Noise	Tap the check box to toggle noise on and off. Set the amount of noise to add to the output signal using the keypad or the multipurpose controls.

AFG output amplitude ranges by impedance (peak to peak)

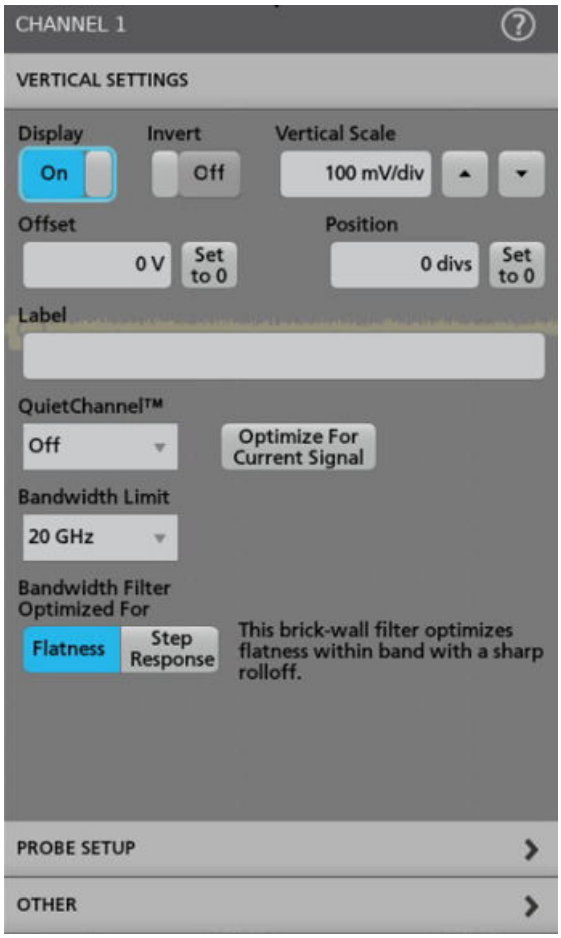
Waveform	50 Ω	High Z
Sine	10 mV to 2.5 V	20 mV to 5 V
Square	10 mV to 2.5 V	20 mV to 5 V
Pulse	10 mV to 2.5 V	20 mV to 5 V
Ramp	10 mV to 2.5 V	20 mV to 5 V
DC	0 V to ±1.25 V	0 V to ±2.5 V
Noise	10 mV to 2.5 V	20 mV to 5 V
Sine(x)/x	10 mV to 1.5 V	20 mV to 3 V
Gaussian	10 mV to 1.25 V	20 mV to 2.5 V
Lorentz	10 mV to 1.2 V	20 mV to 2.4 V
Exponential Rise	10 mV to 1.25 V	20 mV to 2.5 V
Exponential Decay	10 mV to 1.25 V	20 mV to 2.5 V
Haversine	10 mV to 1.25 V	20 mV to 2.5 V
Cardiac	10 mV to 2.5 V	20 mV to 5 V
Arbitrary	10 mV to 2.5 V	20 mV to 5 V

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 for the channel.

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

Vertical Settings panel, fields, and controls



Field or control	Description
Display	Enables **(On)** or disables **(Off)** displaying the channel waveform in the Waveform View window.
Invert	Reverses the amplitude polarity of each sample point. When On, Invert adds a down arrow symbol to the channel badge. See Invert channel guidelines (on page 200).
Vertical Scale	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Offset	Sets the channel signal vertical offset. When there is an offset, the channel handle displays it in the waveform view.
Set to 0	Sets the channel signal vertical offset to 0 units.
Position	Double-tap the field to set the vertical position using the virtual keypad.
Set to 0	Tap to set the waveform zero volt level to the center of the slice or waveform view.
Label	Double-tap the field to add a label to the channel display using the virtual keypad.
Quiet Channel	Select the Quiet Channel setting from the drop-down list as **QC1** - **QC7** or **Off**. Default is off. Refer QuietChannel technology guidelines (on page 198).

Field or control	Description
Optimize for Current Signal	Tap to execute an algorithm that optimizes the Quiet Channel setting based on signal currently being driven into the channel.
Bandwidth Limit	Tap to select the bandwidth limit from the drop-down list.
Bandwidth Filter Optimized For	Tap to select a bandwidth filter that is optimized for flatness or step response. **Flatness** 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. **Step Response** selects a Bessel-Thompson filter that minimizes overshoot with a gradual rolloff.
Other channel settings panels	
Probe Setup	Use to see probe information, check the probe compensation status, compensate the probe, or restore the factory defaults. See Probe Setup panel (Channel configuration menu) (on page 200). See Probe Compensation configuration menu (analog channels Probe Setup panel) (on page 202). See Unsupported Probe dialog (on page 201).
Other	Use to adjust signal delay to align signal arrival at the oscilloscope between probes and/or cables. See Other panel (Channel configuration menu) (on page 202).

QuietChannel™ technology guidelines

QuietChannel™ technology optimizes the oscilloscope dynamic range versus frequency based on the input signal frequency content. It reduces oscilloscope noise and further opens the eye for high-speed serial applications by boosting higher frequency components of the analog signal to reduce noise both ahead of the preamp and after the preamp, but before the ADC. This allows the feature to effectively act as a CTLE function and enables visibility of higher harmonics.

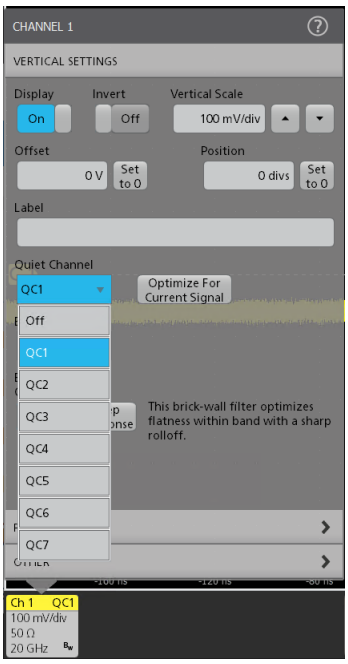
QuietChannel™ technology is intended for use primarily with signals that suffer from high channel loss at higher frequencies. QuietChannel™ technology offers the greatest benefit when acquiring the signal at the far end of the channel from the transmitter, after it has suffered channel loss and the high frequency content has been attenuated. QuietChannel™ technology will boost the high-frequency content of the signal ahead of the ADC, so that it can be digitized with the greatest dynamic range and minimal noise.

The benefit of QuietChannel™ technology will generally be limited when acquiring the signal at the near end, directly at the transmitter, and may actually reduce oscilloscope performance. This occurs because QuietChannel™ Technology will still boost the high frequency signal content, even though it has not been attenuated by a channel. To digitize this boosted signal, the vertical range of the oscilloscope will need to be increased to avoid vertical clipping. The higher vertical scales generally have a higher noise floor thus the noise in the acquired signal will be increased, nullifying the benefit of QuietChannel™ technology. When using Quiet Channel™ technology, the correct balance between amount of boost and vertical range must be found. An automatic optimization function has been included to help you determine the optimal settings.

QuietChannel™ technology access and behaviors

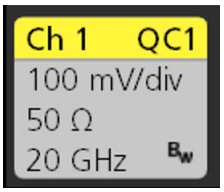
Settings to configure QuietChannel™ technology are available from the Vertical Settings menu.

quiet channel settings



- The Quiet Channel settings can be selected from the drop-down. QC1-7 or Off can be selected.
- Each setting from QC1 to QC7 provides an increasing level of boost.
  - QC1-4 boosting peaks in the 9-12 GHz range. However, these settings create additional loss above 20 GHz. If the oscilloscope bandwidth is set to a value > 20 GHz and QC1-4 are selected, the oscilloscope bandwidth will automatically be limited to a maximum of 20 GHz.
  - QC5-6 boosting peaks in the 16-20 GHz range.
- **Optimize for Current Signal** button finds the optimal Quiet Channel setting, vertical scale, and offset for a given input signal.
  - The optimization routine is designed for high-speed serial signals and searches for the Quiet Channel settings that provides the largest eye opening.
  - If the bandwidth limit is set to >20 GHz, the optimize routine will only select a Quiet Channel setting from QC5-7.
  - If the bandwidth limit is set to 20 GHz or less, the optimization routine will select a quiet channel setting from QC1-7.
- The Quiet Channel setting stays the same if the user changes vertical scale, position, or offset.
- Vertical scale, position, and offset do not change if the Quiet Channel setting is manually changed.
- The channel badge indicates which Quiet Channel setting is in use when QuietChannel™ technology is enabled.

QC2.png



Using QuietChannel™ technology

1. Connect the oscilloscope to a DUT.
2. Double-tap the channel badge to open the Vertical Settings menu.
3. Tap **Optimize for Current Signal** button to start the procedure to determine the optimal Quiet Channel setting given the characteristics of the your DUT's waveform.

Invert channel guidelines

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

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

- The instrument clears acquisitions and restarts triggering
- All persistence information resets/clears, including all waveform statistics, etc
- All associated views and derived results change. For example, XY view, cursor readouts, measurement results, search results, math, bus decode, and so on, all change to reflect the current Invert setting

Settings that change when Invert is set to On:

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

Settings that don't change when Invert is On:

- The channel **Vertical** scale does not change
- The channel **Vertical** position knob rotation direction does not change
- The channel Vertical **Position** value does not change
- The **Trigger** level knob rotation direction does not change

Invert and Setup, Autoset, and Default functions

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

Probe Setup panel (Channel configuration menu)

Use the Channel configuration menu Probe Setup panel to see probe information, check the probe compensation status, compensate the probe, or restore the factory defaults.

To open the Probe Setup panel:

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

Probe Setup panel fields and controls

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

Field or control	Description
Probe Information	View probe information such as probe type, serial number, and its attenuation.
Probe Compensation Status	View the probe compensation status: Default, Pass, or Fail.

Field or control	Description
Compensate Probe	Tap to display the probe compensation dialog. This is only available for probes that support automatic compensation.
Mode	Select the probe operating mode from the drop down list (Differential, Common Mode, A, or B).
Offset	Enter the A or B offset value. This value is the same as the Offset in the Settings panel.
Differential Offset Tracking	When checked, the Differential Offset Tracking mode is Tracking. When unchecked, the Differential Offset Tracking is Manual.
Differential Offset	When in manual mode, enter the Differential Offset value.
Common Mode Offset	When in manual mode, enter the Common Mode Offset value.
Common Mode Offset Tracking	When checked, the Common Mode Offset Tracking mode is Tracking. When unchecked, the Common Mode Offset Tracking is Manual.
VTerm Tracking	When checked, the VTerm mode is Tracking. When unchecked, the VTerm is Manual.
VTermA	When in manual mode, enter the VTermA value.
VTermB	When in manual mode, enter the VTermB value.

Probe error messages are also displayed in this setup panel, see Badges (on page 46).

See also

Unsupported Probe dialog (on page 201)

Probe Compensation configuration menu (analog channels Probe Setup panel) (on page 202)

Other panel (Channel configuration menu) (on page 202)

Deskew configuration menu (Other panel, Channel configuration menu) (on page 92)

Unsupported Probe dialog

This dialog tells you that you have attached an unsupported probe

If the instrument detects an unsupported probe has been attach, the Unsupported Probe dialog appears.

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 202)

Other panel (Channel configuration menu) (on page 202)

Deskew configuration menu (Other panel, Channel configuration menu) (on page 92)

Probe Compensation configuration menu (analog channels Probe Setup panel)

Use this menu to compensate probe. This menu is only available when a compensation-supported probe is installed on the channel.

To open the Probe Compensation dialog:

1. Double-tap the Channel badge on the Settings bar to open the channel configuration menu.
2. Tap the **Probe Setup** panel.
3. Tap **Compensate Probe**.

Probe Compensation dialog

Available fields and controls vary with the type of probe that is attached. For more information, consult the probe documentation. Read the information on the menu before starting the probe compensation process.

Field or control	Description
Compensate Probe	Tap to compensate the attached probe. Before compensating the probe, read the instructions in the dialog.
Restore Factory Defaults	Tap to restore the probe compensation factory defaults and remove the previous compensation results.
Probe Compensation Status	Probe compensation status can be Running, Passed, Failed or Default.

See also

Unsupported Probe dialog (on page 201)

Other panel (Channel configuration menu) (on page 202)

Deskew configuration menu (Other panel, Channel configuration menu) (on page 92)

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

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

Field or control	Description
Deskew	Sets or displays the probe deskew value.
Set to 0	Sets the probe deskew value to zero (0) seconds.

Field or control	Description
Multi-Channel	Opens a Deskew configuration menu that allows you to deskew multiple channels (two at a time). You can set the deskew method to propagation delay (default) or measured to use an algorithm.
External Attenuation	Double-tap the numeric fields to set the external attenuation using the virtual keypad.
Set to Unity	Sets the external attenuation to unity.
Alternate Units	Toggles Alternate Units On or Off. Use to set custom vertical measurement units. The vertical scale will show the entered measurement units.
Units	Double-tap to enter alternate units using the virtual keyboard. Only shown when **Alternate Units** = **On**.
Ratio	Double-tap the numeric fields to set the ratio using the virtual keypad. Only shown when **Alternate Units** = **On**.
Set to Unity	Sets the ratio to unity. Only shown when **Alternate Units** = **On**.

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.

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 (on page 73).

Deskew menu fields and controls

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

Field or control	Description
From Source	Tap and select from the drop-down list the channel to deskew from (your reference channel for deskewing).
To Source	Tap and select from the drop-down list the channel to deskew to (the channel that you want to match the **From Source** reference channel).

Field or control	Description
Probe	If the oscilloscope recognizes the probe attached to the channel, the **Probe** field shows the nomenclature of the attached probe. If the oscilloscope does not recognize the probe attached to the channel, the **Probe** field shows a drop-down list from which you can select the probe that is attached to the selected channel. If the attached probe is not in the list, select **Custom** (at the bottom of the list) and enter the probe propagation delay in the **Prop. Delay** field.
Prop. Delay	Field in which you can enter the probe delay for an unsupported probe. Double-tap in the field and use the keypad to enter the propagation delay value. This field is available when **Probe** = **Custom**.
Propagation Delay	This field lists the default propagation delay of the attached probe. A positive value shifts a channel to the left. When this button is selected and probes attached, the propagation delay is automatically detected and shows on the right.
Measured	This button selects a deskew method that applies an algorithm to automatically adjust channel timing alignment.
OK, deskew	Sets the oscilloscope to add or subtract the delay values of the **To Source** channel such that the delay between the two channels is as close to 0 as possible.

Bus configuration menu

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

To open the Bus configuration menu:

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

Bus configuration menu - fields and controls

Field or control	Description
Display	Toggles bus display **On** or **Off**.
Label	Enter label text in this field. The default label is the name of the bus type.
Position	Sets the vertical position of the bus waveform. Default is 0 (center of graticule).
Set to 0	Sets the vertical position of the bus waveform to 0 (center graticule).
Bus Type	Select a bus from the drop down list. The Parallel bus type comes standard on the instrument. Serial buses require purchase and installation of serial bus triggering and analysis options.
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.

Field or control	Description
Display format	Enables showing just the decoded bus or both the bus and its digital waveforms. You can also tap on the + symbol on the bus waveform to toggle between showing the bus only or showing bus and source waveforms.
Decode format	Sets how decoded data information is shown in the bus. Select from listed formats. Available formats depend on the bus type.

128b132b serial bus configuration menu

Use the 128b132b bus menu to set up and display a 128b132b waveform.

To set up a 128b132b serial bus:

1. To create a new 128b132b bus waveform, tap the **Add New Bus** button on the Settings bar. Double-tap the badge to open the bus configuration menu. Set the **Bus Type** to **128b132b**.
2. To change the settings on an existing 128b132b serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu.

128b132b serial bus menu fields and controls

Field or control	Description
Display	Enables or disables the display of the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **128b132b**.
Source	Select the source channel from the drop-down menu.
Threshold	Sets the threshold value for the signal source. The value range is -8 V to +8 V.
Bit Rate	Sets the data bit rate. The value range is 1 bps to 25 Gbps.
Bit Order	Set the bit order for the bit as **LSB** or **MSB**.
Symbol Size	Select the symbol size from drop-down list. Available symbol sizes are Nibble, 1 byte, 2 bytes, 4 bytes, and 8 bytes.
Symbols	Enter the symbols value according to symbol size. For Nibble, the symbols range from 8 to 32. For 1 byte, the symbols range from 4 to 16.

Field or control	Description
Pattern	Enter the pattern. Tap the Pattern field and use the A and B knobs to select and change the values. You can also double-tap on the field to use the virtual keypad to enter values.
Display Format	Sets the waveform view to show just the decoded bus information. **Bus** sets the waveform view to display just the decoded bus information. **Bus and Waveform** sets the waveform view to display both the decoded bus and the source signal waveforms. You can also tap on the + symbol on the bus waveform to toggle between show the bus only or showing bus and source waveforms.
Decode Format	Sets the decode format for the bus information.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu.

See also

Bus Search configuration menus (on page 375)

128b132b serial bus search configuration menu (on page 375)

1-Wire serial bus configuration menu

Use the 1-Wire bus menu to set up and display an 1-Wire serial bus waveform.

To set up the 1-Wire serial bus:

- To add a new 1-Wire 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 **1-Wire**.
- To change the settings on an existing 1-Wire serial bus, double-tap the 1-Wire **Bus** badge and make necessary changes.

1-Wire serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter the label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter the text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **1-Wire**.

Field or control	Description
Mode	Selects the standard. Available options are **Standard (15.4 Kbps)** and **Overdrive (125 Kbps)**. Default is **Standard (15.4 Kbps)**.
Source	Select the channel source for the signal.
Threshold	Select the threshold for the signal.
Display Format	Sets the waveform view to show just the decoded bus information.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Mixed Hex**, and **Binary**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

1-Wire serial bus search configuration menu (on page 376)

64B/66B serial bus configuration menu

Use the 64B/66B bus menu to set up and display a 64B/66B waveform.

To set up a 64B/66B serial bus:

1. To create a new 64B/66B bus waveform, tap the **Add New Bus** button on the Settings bar. Double-tap the badge to open the bus configuration menu. Set the **Bus Type** to **64B/66B**.
2. To change the settings on an existing 64B/66B serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu.

64B/66B serial bus menu fields and controls

Field or control	Description
Display	Enables or disables the display of the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).

Field or control	Description
Bus Type	Set to **64B/66B**.
Source	Select the source channel from the drop-down menu.
Threshold	Sets the threshold value for the signal source. The value range is -8 V to +8 V.
Bit Rate	Sets the data bit rate. The value range is 1 bps to 25 Gbps.
Bit Order	Set the bit order for the bit as **LSB** or **MSB**.
Symbol Size	Select the symbol size from drop-down list. Available symbol sizes are Nibble, 1 byte, 2 bytes, 4 bytes, and 8 bytes.
Symbols	Enter the symbols value according to Symbol Size. For Nibble, the symbols range from 8 to 16. For 1 byte, the symbols range from 4 to 8.
Pattern	Enter the pattern. Tap the Pattern field and use the A and B knobs to select and change the values. You can also double-tap on the field to use the virtual keypad to enter values.
Display Format	Sets the waveform view to show just the decoded bus information. **Bus** sets the waveform view to display just the decoded bus information. **Bus and Waveform** sets the waveform view to display both the decoded bus and the source signal waveforms. You can also tap on the + symbol on the bus waveform to toggle between show the bus only or show bus and source waveforms.
Decode Format	Sets the decode format for the bus information.

Other bus types

Other serial bus types are available as purchasable options. Once purchased and installed, the new bus types are shown in the Bus Type menu.

See also

Bus Search configuration menus (on page 375)

64B/66B serial bus search configuration menu (on page 377)

8b10b serial bus configuration menu

Use the 8b10b bus menu to set up and display an 8b10b serial bus waveform.

To set up a 8b10b serial bus:

- To add a new 8b10b bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double-tapping the badge. Set the **Bus Type** to **8b10b**.
- To change the settings on an existing 8b10b serial bus waveform, double-tap the **8b10b** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

You can configure search for 8b10b (on page 378) and define conditions to search for and mark on an 8b10 bus waveform.

8b10b serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **8b10b**.
Source	Select the signal source from listed analog channels.
Threshold	Sets the threshold value for the signal source.
Bit Rate	Set the bit rate. You can configure values from 1 bps to 1 Gbps.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and Symbolic Decode.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

8b10b serial bus search configuration menu (on page 378)

ARINC 429 serial bus menu

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

To set up the ARINC 429 avionics serial data bus:

- Tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the new Bus badge. Set the **Bus Type** to **ARINC429**.
- To change the settings on an existing ARINC 429 serial bus waveform, double-tap the **Bus** waveform badge and make necessary changes.

ARINC 429 serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text using an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **ARINC429**.
Polarity	Select the polarity to match the ARINC 429 bus being acquired.
Source	Select the ARINC 429 signal source.
High Threshold, Low Threshold	Sets the valid high and low threshold values for the signal source.
Bit Rate	Sets the bit rate to 12,500, 100,000, or Custom.
Custom Rate	Sets a custom data bit rate. To set the value, tap the field and use the Multipurpose knob, double-tap the field and use the Custom Rate virtual keypad, or double-tap the field and use an attached keyboard. Only available when Bit Rate = Custom.
Data Format	Sets the data format to Data (19 bits), SDI (Source/Destination Identifiers) plus data (21 bits), or SDI plus Data plus Sign/Status Matrix (SSM) (23 bits.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and Mixed Hex.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

ARINC 429 serial bus search configuration menu (on page 380)

Audio serial bus configuration menu

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

To set up the Audio serial bus:

- To create a new Audio bus waveform, tap the **Add New Bus** button on the Settings bar and open the Bus configuration menu. Set the **Bus Type** to **Audio**.
- To change the settings on an existing Audio serial bus waveform, double-tap the **Bus** waveform badge and make the necessary changes.

Audio serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **Audio**.
Audio Type	Sets the digital audio signal type. Select from the drop-down list.
Bit Order	Set the waveform to decode with most-significant (MS) bit first or least-significant (LS) bit first.
Bit Clock	Set the signal source, logic level threshold, and polarity for the Bit Clock signal.
Word Select	Set the signal source, logic level threshold, and normal or invert signal setting for the Word signal.
Data	Set the signal source, logic level threshold, and logic definition (active high or low) for the Data signal.
Word Size (Audio Type = I2S, LJ, or RJ)	Set the number of bits used in a Word for the selected audio type (8, 12, 16, 18, 20, 24, 28, or 32-bits).
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information.
TDM-specific settings	

Field or control	Description
Frame Sync	Set the signal source, logic level threshold, and polarity for the frame sync signal.
Data Bits per Channel	Set the number of data bits per audio channel.
Clock Bits per Channel	Set the number of clock bits per audio channel.
Channels per Frame	Set the number of audio channels per data frame.
Bit Delay	Sets the bit delay (number of bits).

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

Audio serial bus search configuration menu (on page 379)

Auto Ethernet serial bus configuration menu

Use the Auto Ethernet bus menu to set up and display an Auto Ethernet serial bus waveform.

To set up a Auto Ethernet serial bus:

- To add a new Auto Ethernet serial 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 **Auto Ethernet**.
- To change the settings on an existing Auto Ethernet serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.



You can configure search for Auto Ethernet (on page 382) and define conditions to search for and mark on an Auto Ethernet bus waveform.

Auto Ethernet serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions centered vertically in a slice or on the screen.
Bus Type	Set to **Auto Ethernet**.
Speed	Sets the network signal speed as **10 Base-T1S, 100 Base-T1,** or **1000 Base-T1**.
Mode	Sets the mode to **Training** or **Data**. Mode is available only when speed is set to **1000 Base-T1**.
Control Role	Sets the control role to **Master** or **Slave**.
Packet View	Turns on or off the packet view of the bus in the waveform view. Packet view is available only when **Speed** is set to **10 Base-T1S** or **1000 Base-T1**.
Threshold	Sets the threshold value for the signal source. Threshold is available when **Speed** is set to **10 Base-T1S** and for **1000 Base-T1**, when mode is set to **Training**.
Bit Rate	Sets the bit rate. Bit Rate is available when **Speed** is set to **10 Base-T1S**.
Data Rate	Sets the data rate. Data Rate is available when **Speed** is set to **1000 Base-T1** and mode is set to **Training**.
Use RS-FEC	Select the **Use RS-FEC** box to enable the error correction. **Use RS-FEC** is available when speed is set to **1000 Base-T1** and mode is set to **Data.**
Signal type	Sets the signal type to **Single-Ended** or **Differential**.
Source	Select the channel source for the signal from a differential probe. Source is available when **Signal Type** is set to **Diff**.

Field or control	Description
High Threshold	Sets the threshold value for a logic 1 value. High Threshold is available when **Signal Type** is set to **Diff**.
Low Threshold	Sets the threshold value for a logic 0 value. Low Threshold is available when **Signal Type** is set to **Diff**.
D+ Input, D- Input	Sets the signal sources and threshold values for the Single Ended data+ and - signals. Available when **Signal Type** is set to **Single-Ended**.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, MixedHex, and Binary.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

Auto Ethernet serial bus search configuration menu (on page 382)

CAN serial bus configuration menu

Use the CAN bus menu 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
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. Default standard 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.

Field or control	Description
Position	Sets the vertical position of the bus waveform. Default standard position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **CAN** to set up and display a CAN bus waveform.
Signal Type	Sets the CAN signal type to decode. Available signal types are Diff, CAN_H, CAN_L, Rx, and Tx. Default standard is **CAN_H**.
CAN Standard	Sets the CAN signal standard to decode. Available CAN standards are CAN 2.0, CAN FD (ISO), CAN FD (non-ISO), CAN 2.0 + FD (ISO) + XL, and CAN XL. Default standard is **CAN 2.0**.
Source	Select the signal source from listed analog channels.
Threshold	Sets the high/low logic transition level.
Sample Point	Sets the sample point from 5% to 95% of the position within the bit period or the unit interval.
Bit Rate	Sets the bit rate of your CAN bus serial data. To enter a custom bit rate, select **Custom** and enter the custom bit rate in the **Custom Rate** input box. This control is available when **CAN Standard** is set to **CAN 2.0**.
SD Bit Rate	Sets the SD (Standard/Arbitration) bit rate of your CAN FD serial bus data. To enter a custom bit rate, select **Custom** and enter the custom bit rate in the **Custom Rate** input box. This control is available when **CAN Standard** is set to **CAN FD (ISO)**, **CAN 2.0 + FD (ISO) + XL**, or **CAN FD (non-ISO)**. **NOTE:** Calculating SD bit rate of the waveform requires placing two cursors on either side of the smallest time interval section of low frequency part of the waveform which will give us the SD bit rate (1/Δt).
FD Bit Rate	Sets the FD (Faster Data) bit rate of your CAN FD serial bus data. To enter a custom bit rate, select Custom and enter the custom bit rate in the Custom Rate input box. This control is available when **CAN Standard** is set to **CAN FD (ISO)**, **CAN 2.0 + FD (ISO) + XL**, or **CAN FD (non-ISO)**. **NOTE:** Calculating FD bit rate of the waveform requires placing two cursors on either side of the smallest time interval section of high frequency part of the waveform which will give us the FD bit rate (1/Δt).

Field or control	Description
Nominal Bit Rate	Sets the data rate for Arbitration mode. To enter a custom bit rate, select **Custom** and enter the custom bit rate in the **Custom Rate** input box. Ranges from 10 Kbps to 1 Mbps. This control is available when **CAN Standard** is set to **CAN XL**.
XL Bit Rate	Sets the data rate for Data Transfer mode. To enter a custom bit rate, select **Custom** and enter the custom bit rate in the **Custom Rate** input box. This control is available when **CAN Standard** is set to **CAN XL** or **CAN 2.0 + FD (ISO) + XL**. **NOTE:** Calculating XL bit rate of the waveform requires placing two cursors on either side of the smallest time interval section of high frequency part of the waveform which will give us the XL bit rate (1/Δt).
Custom rate	Sets the custom bit rate to use to decode the signal. Tap the field and use the Multipurpose knob to change the value, or double-tap on the field and use the virtual keypad to enter a custom bit rate. This control is available when **Bit Rate**, **SDI Bit Rate**, **Nominal Bit Rate**, **SD Bit Rate**, **FD Bit Rate**, **Nominal Bit Rate**, or **XL Bit Rate** is set to **Custom.**
Display Format	**Bus** sets the Waveform view to show just the decoded bus information. **Bus and Waveform** sets the Waveform view to show both the decoded bus and the source signal waveforms. You can also tap on the **+** symbol on the bus waveform to toggle between showing the bus only or showing bus and source waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, **Mixed Hex**, and **Symbolic**. Selecting **Symbolic** 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. **NOTE:** .dbc files are not saved as part of a save session operation. To remove the .dbc symbolic information from the waveform, select a different **Decode Format**.
Browse	Opens the standard **File Open** dialog to navigate to and load a .dbc file to apply to the bus being created. You can navigate to TekDrive to load .dbc file. The **File Open** dialog automatically filters for the .dbc file extension. Available when **Decode Format** is set to **Symbolic**.
Path field	Use to enter the path to a .dbc file, or select from a drop-down list of recently accessed .dbc files. Available when **Decode Format** is set to **Symbolic**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

CPHY serial bus configuration menu

Use the CPHY bus menu to set up and display an CPHY serial bus waveform.

To set up the CPHY serial bus:

- To add a new CPHY 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 **CPHY**.
- To change the settings on an existing CPHY serial bus, double-tap the CPHY **Bus** badge and make necessary changes.

CPHY serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter the label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter the text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **CPHY**.
Sub Type	Sets the sub type. Available options are **CSI**, **DSI**, **Word**, **Symbol**.
Signal Type	Sets the signal type. Available options are **Single Ended** and **Diff**.
Bit Rate	Sets the bit rate.
Source	Select the channel source for the signal.
Threshold	Select the threshold for the signal.
LP Direction	Sets the LP direction to forward or reverse. Available only when **Sub Type = DSI**

Field or control	Description
LP Threshold	Select the LP threshold for the signal. Available only when **Signal Type = Single Ended**
Display Format	Sets the waveform view to show just the decoded bus information.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Mixed Hex**, and **Binary**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

CPHY serial bus search configuration menu (on page 390)

CXPI serial bus configuration menu

Use the CXPI bus menu to set up and display an CXPI serial bus waveform

To set up the CXPI serial bus:

- To add a new CXPI 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 **CXPI**.
- To change the settings on an existing CXPI serial bus, double-tap the CXPI **Bus** badge and make necessary changes.

CXPI serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter the label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter the text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **CXPI**.
Bit Rate	Sets the bit rate to decode the waveform.
Data Input	Sets the source and threshold (recessive) level for the signal.

Field or control	Description
Display Format	Sets the waveform view to show just the decoded bus information.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Mixed Hex**, and **Binary**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

CXPI serial bus search configuration menu (on page 393)

DPHY serial bus configuration menu

Use the DPHY bus menu to set up and display an DPHY (clock, data + and data -) serial bus waveform.

To set up the DPHY serial bus:

- To add a new DPHY 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 **DPHY**.
- To change the settings on an existing DPHY serial bus, double-tap the DPHY **Bus** badge and make necessary changes.

DPHY serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **DPHY**.
Protocols	Select the protocol to decode. Available options are **CSI** and **DSI**. Default is CSI.

Field or control	Description
LP Direction	Set the LP (Low Power) direction, host to peripheral or peripheral to host. Available options are **Forward** and **Reverse**. Available only when **Protocols** = **DSI**.
Source	Select the channel source for the signal.
Data Threshold	Select the data threshold for Dp and Dn signals.
LP Threshold	Select the LP threshold for Dp and Dn signals.
Dp Input, Dn Input	Sets the signal sources.
8b9b Encoding	Turns on or off 8b9b line encoding to support additional control features. Default is OFF.
Display Format	Sets the waveform view to show just the decoded bus information.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Mixed Hex**, and **Binary**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

DPHY serial bus search configuration menu (on page 403)

eSPI serial bus configuration menu

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

To set up the eSPI serial bus:

- To add a new eSPI 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 **eSPI**.
- To change the settings on an existing eSPI serial bus, double-tap the eSPI **Bus** badge and make necessary changes.

eSPI serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter the label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter the text from an attached keyboard.

Field or control	Description
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **eSPI**.
I/O Mode	Toggles between **Single** and **Dual** mode.
Alert	Turns on or off the alert signal.
SCLK Input	Select the channel source and threshold level for the Serial Clock signal. Set the Polarity to rising or falling edge of the clock signal used by the master device to start transferring bits.
CS Input	Select the channel source and threshold level for the Chip Select signal. Set the Polarity to use Active High or Active Low logic for the CS signal.
Command Input	Select the channel source and threshold level for the command input signal. Set the Polarity to use Active High or Active Low logic for the command signal. Available only when **I/O mode = Single**.
Response Input	Select the channel source and threshold level for the response input signal. Set the Polarity to use Active High or Active Low logic for the response signal. Available only when **I/O mode = Single**.
IO[0]Input	Select the channel source and threshold level for the IO[0] signal. Available only when **I/O mode = Dual**.
IO[1]Input	Select the channel source and threshold level for the IO[1] signal. Available only when **I/O mode = Dual**.
Display Format	Sets the waveform view to show the decoded bus information.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Mixed Hex**, and **Binary**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

eSPI serial bus search configuration menu (on page 405)

Ethernet serial bus menu

Use the Ethernet bus menu to set up and display an Ethernet 10 BaseT 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
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **Ethernet**.
Speed	Sets the network signal speed.
Signal Type	Sets the signal type to Single Ended or Differential.
Source	Sets the signal source for Differential. Only available when Signal Type = Diff.
D+ Input, **D- Input**	Defines the signal sources and threshold values for the Single Ended data+ and - signals. Only available when Signal Type = Single Ended.
High Threshold	Sets the threshold value for a logic 1 value. Only available when Signal Type = Diff.
Low Threshold	Sets the threshold value for a logic 0 value. Only available when Signal Type = Diff.
IPv4	Set to Yes if the Ethernet signal being measured uses Internet Protocol version 4 (IPv4).

Field or control	Description
Q-(VLAN)	Set to Yes if the Ethernet signal being measured uses IEEE 802.1Q virtual LANs.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, Mixed ASCII, and Mixed Hex.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

Ethernet serial bus search configuration menu (on page 412)

EtherCAT serial bus configuration menu

Use the EtherCAT bus menu to set up and display an EtherCAT serial bus waveform.

To set up a EtherCAT serial bus:

- To create a new EtherCAT 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 **EtherCAT**.
- To change the settings on an existing EtherCAT serial bus waveform, double-tap the EtherCAT **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

EtherCAT serial bus menu fields and controls

Field or control	Description
Display	Enables or disables displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (Stacked mode), or center screen (Overlay mode). The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **EtherCAT**.
Signal Type	Set the signal type to **Single Ended** or **Diff**.

Field or control	Description
D+ Input, D- Input	Defines the signal sources and threshold values for the Single Ended data + and - signals. Available only when **Signal Type** is **Single Ended**.
Source	Set the signal source as **Diff** for differential source. Available only when **Signal Type** is **Diff**.
Threshold	Set the threshold value for the differential signal. Available only when **Signal Type** is **Diff**.
Display Format	Set the waveform view to show the decoded bus information.
Decode Format	Sets the decode format used to display the bus information. Available formats are Hex, Binary, and Mixed Hex.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

EtherCAT serial bus search configuration menu (on page 414)

eUSB serial bus configuration menu

Use the Embedded USB bus menu to set up and display an eUSB 2.0 (Embedded Universal Serial Bus) waveform.

To set up a eUSB serial bus:

- To create a new eUSB 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 **eUSB**.
- To change the settings on an existing eUSB serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

eUSB serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.

Field or control	Description
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **eUSB**.
Speed	Set the speed to match the acquiring speed of the eUSB bus. Available data rate options are **Full (12 Mbps)**, **High Speed (480 Mbps)**, and **Low (1.5 Mbps)**.
Mode	Set the mode of operation to Native or Repeater. Default is Native. ▪ **Native** mode is used for inter chip disconnect. ▪ **Repeater** is used to communicate eUSB2 port with USB2.0 through repeater that translates between 2USB2 signaling and USB2.0 signaling. ▪ **Repeater Peripheral**
Signal Type	Set to match the eUSB signal you are acquiring (Single Ended or Differential). Use a differential probe to acquire the differential USB signal. Both Single Ended and Diff can be used to measure High speed USB signals. Single Ended is used to measure Full (12 Mbps) and Low (1.2 Mbps) speed USB signals. **NOTE:** Signal Type selection is applicable only in High Speed.
Source	Select the channel source for the signal from a differential probe. Only available when **Signal Type** = **Diff**.
Threshold	Set the threshold level.
Data Threshold	Set the data threshold for D+ signal and D- signal. Only available when **Signal Type** = **Single Ended**.
High Threshold	Set the high threshold for D+ signal and D- signal.
Low Threshold	Set the low threshold for D+ signal and D- signal.
D+ Input	Select the channel source and set the threshold level for the Data+ signal.
D- Input	Select the channel source and set the threshold level for the Data- signal.
Display Format	Sets the waveform view to show the decoded bus information or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, **Mixed ASCII**, and **Mixed Hex**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

eUSB serial bus search configuration menu (on page 418)

FlexRay serial bus configuration menu

Use the Flexray bus menu 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
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text using an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **FlexRay**.
Signal Type	Select the FlexRay signal type being measured.
Channel Type	Set to A or B channel.
Source	Select the FlexRay signal source.
Threshold	Sets the threshold value for the TX or RX signal type.
High Threshold, Low Threshold	Sets the high and low threshold values for the BM Inverted and Bdiff/BP signal types.
Bit Rate	Select a bit rate. To set a custom bit rate, select Custom and enter a value in the Custom Rate field.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

FlexRay serial bus search configuration menu (on page 422)

I2C serial bus configuration menu

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

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

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

I2C serial bus menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **I2C**.
SCLK Input	Sets the source and threshold level for the Serial Clock Line signal.
SDA Input	Sets the source and threshold level for the Serial Data signal.
Include R/W bit in Address	Select **Yes** to display 7-bit addresses as eight bits, where the eighth bit (LSB) is the R/W bit, or display 10-bit addresses as 11 bits, where the third bit is the R/W bit. Select **No** to display 7-bit addresses as seven bits, and 10-bit addresses as ten bits.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

I2C serial bus search configuration menu (on page 424)

I3C serial bus configuration menu

Use the I3C bus menu to set up and display a decoded I^3C serial bus waveform.

To set up the I^3C serial bus menu:

- To create a new I^3C 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^3C serial bus waveform, double-tap the I^3C **Bus** waveform badge and make necessary changes in the configuration menu.

I3C serial bus menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	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.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in **Stacked** mode), or center screen in **Overlay** mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **I3C**.
Version	Sets the version as **1.0** or **1.1** for I3C bus.
SCLK Input	Sets the source and threshold level for the Serial Clock Line signal.
SDA Input	Sets the source and threshold level for the Serial Data signal.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

I3C serial bus Search configuration menu (on page 425)

LIN serial bus configuration menu

Use this menu 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

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **LIN**.
Source	Set the signal source from available analog channels.
Threshold	Set the threshold level to define a logic high level.
Polarity	Select the polarity to match the LIN bus being acquired.
LIN Standard	Select the standard to match the LIN bus being acquired.
Bit rate	Sets the bit rate. To enter a custom bit rate, tap Custom and enter the custom bit rate in the Custom Rate input box.
Include Parity Bits with ID	Set to Yes to include parity bits with the ID.
Sample Point	Sets the sample point from 5% to 95% of the position within the bit period or the unit interval.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and Mixed.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

LIN serial bus search configuration menu (on page 436)

Manchester serial bus configuration menu

Use the Manchester bus menu to set up and display a manchester waveform.

To set up a Manchester serial bus:

- To create a new Manchester 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 **Manchester**.
- To change the settings on an existing Manchester serial bus, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

Manchester serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus decode. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **Manchester**.
Set the edge for bit '0'	Set the edge for bit 0.
Data Rate	Set the data rate for the manchester bus. Default data rate is 125 kb/s.
Source	Select the signal source.
Threshold	Set the threshold level.
Start Index	Select the start index. Default is one edge. Unit is edge.
Packet View	Set to **On** to show packet field level decode and **Off** show bit level decode. Default is ON.
Idle Bits	Set the idle bits depending on the idle period of time. Default is 1.2.

Field or control	Description
Tolerance	Set the tolerance for giving bandwidth to edge in the bit for bit error. Default is 10 %.
Sync Bits	Set the number of bits that define sync pattern for the manchester bus. Default is 1 bit. Available only when **Packet View** is **On**.
Parity	Set the manchester decode in the packet from the waveform being acquired. Default is None. Available only when **Parity View** is **On**.
Bit Order	Set the order of bit to Most Significant Bit first (MSB) or Least Significant Bit first (LSB). Default is MSB. Available only when **Packet View** is **On**.
Word Count	Set the number of words to be decoded in a packet. Default is 1. Available only when **Packet View** is **On**.
Word Size	Set the number of bits in each payload. Default is 8 bits. Available only when **Packet View** is **On**.
Display Format	Sets the waveform view to show the decoded bus information. Format is **Bus**.
Header	Set the number of bits in the header of a packet. Default is 0 bits. Available only when **Packet View** is **On**.
Trailer	Set the number of bits in the trailer of a packet. Default is 0 bits. Available only when **Packet View** is **On**.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex** and **Binary**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

Manchester serial bus search configuration menu (on page 437)

MDIO serial bus configuration menu

Use the MDIO bus menu to set up and display an MDIO (Management Data Input Output) serial bus waveform.

To set up the MDIO serial bus:

- To create a new MDIO 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 **MDIO**.
- To change the settings on an existing MDIO serial bus waveform, double-tap the MDIO **Bus** waveform badge and make necessary changes.

MDIO serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set the **MDIO** to set up and display a MDIO bus waveform.
Clock Input	Set the source and threshold level for the clock input signal.
Data Input	Set the source and threshold level for the data input signal.
Threshold	Set the threshold level to define a logic high level. Sets the threshold value for the signal source.
Display Format	Sets the waveform view to show the decoded bus information, or the decoded bus and the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

MDIO serial bus search configuration menu (on page 438)

MIL-STD-1553 serial bus menu

Use the MIL-STD-1553 bus menu to set up and decode a MIL-STD-1553 aeronautic network serial data bus waveform.

To set up the MIL-STD-1553 serial data bus:

- To create a new MIL-STD-1553 bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **MIL-STD-1553**.
- To change the settings on an existing FlexRay serial bus waveform, double-tap the **Bus** waveform badge and make necessary changes.

MIL-STD-1553 serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text using an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **MIL-STD-1553**.
Polarity	Select the polarity to match the MIL-STD-1553 bus being acquired.
Source	Select the MIL-STD-1553 signal source.
High Threshold, Low Threshold	Sets the valid high and low threshold values for the signal source.
RT Maximum	Sets the maximum valid response time (RT) for a command.
RT Minimum	Sets the minimum valid response time (RT) for a command.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, Mixed ASCII, and Mixed Hex.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

MIL-STD-1553 serial bus search configuration menu (on page 441)

NFC serial bus configuration menu

S Use the NFC bus menu to setup and decode a NFC serial bus waveform.

Follow the procedure to setup the NFC serial bus.

1. Enable the Spectrum View from Channel configuration menu. See Analog Channel configuration menu (on page 196)
2. Configure the Spectrum badge menu. See Spectrum badge menu

   NOTE: The center frequency should be 13.56 MHz.

3. Configure the NFC bus menu.
   - To create a new NFC 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 **NFC**.
   - To change the settings on an existing NFC serial bus waveform, double-tap the NFC **Bus** waveform badge and make necessary changes in the configuration menu.

NFC serial bus menu fields and controls

Field or control	Description
Display	Enables or disables displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter a label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (Stacked mode), or center screen (Overlay mode). The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **NFC**.
Standard	Sets the NFC standard to decode. The default is 15693. The available decode standards are 15693, 14443A, 14443B, and FeliCa.
Data Rate	Set the data rate for the NFC bus. You can configure the data rate from 1 bps to 1000 Mbps. The default rate is 106 kbps for 15693, 14443A, and 14443B. For FeliCa, the default data rate is 212 kbps.
Source	Set the source and threshold level for the NFC input signal.
Command Threshold	Set the command threshold value, refer math waveform with the label CMD for setting the threshold. The command threshold range is -8 V to 8 V.
Response Threshold	Set the response threshold value, refer math waveform with the label RSP for setting the threshold. The response threshold range is -8 V to 8 V.
Transition For '0'	Select the transition for 0 depending on manchester type (IEEE or GE). Available when **Standard** is **15693**, **14443A**, or **FeliCa**.

Field or control	Description
Tolerance	Set the tolerance value. The valid tolerance range is from 1% to 50%. The default tolerance is 10%. Available when **Standard** is **15693**, **14443A**, and **FeliCa**.
Decode Start Edge	Set the decode start edge for the signal. Available when **Standard** is **14443A** or **FeliCa**.
Polarity	Set the polarity as **Normal** or **Inverted**. Available when **Standard** is **14443B**.
Response Load Demod	Tap **Yes** or **No** to apply the demodulation. Available when **Standard** is **14443A** or **14443B**.
Display Format	Set the waveform view to show the decoded bus information.
Decode Format	Set the decode format used to display the bus information. Available formats are Hex, Binary, or Mixed hex.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

NFC serial bus search configuration menu (on page 234)

NRZ serial bus configuration menu

Use the NRZ bus menu to set up and display an NRZ serial bus waveform.

To set up a NRZ serial bus:

- To add a new NRZ bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double-tapping the badge. Set the **Bus Type** to **NRZ**.
- To change the settings on an existing NRZ serial bus waveform, double-tap the **NRZ** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

You can configure search for NRZ (on page 444) and define conditions to search for and mark on an NRZ bus waveform.

NRZ serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.

Field or control	Description
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **NRZ**.
Bit Order	Set the waveform to decode with most-significant (MS) bit first or least-significant (LS) bit first.
Polarity	Select the polarity as **Normal** or **Invert** to match the NRZ bus being acquired.
Source	Select the signal source from listed analog channels.
Threshold	Sets the threshold value for the signal source.
Bit Rate	Set the bit rate. You can configure values from 1 bps to 1 Gbps.
Packet View	Toggles **On** or **Off**, When enabled, it performs per-packet decoding including headers, trailers, and other control elements.
Start Index	Defines the starting edge of the waveform. The default value is 1 edge.
Sync Bits	Sets the number of bits that define sync pattern for the NRZ bus. The default value is 1 bit. Available when **Packet View** is **On**.
Parity	Sets the type of parity as **None**, **Odd**, or **Even**. Use this setting for parity error detection. Available when **Packet View** is **On**.
Word Count	Sets the number of words to be decoded in a packet. The default value is 1. Available when **Packet View** is **On**.
Word Size	Sets the number of bits per word. The default is 8 bits. Available when **Packet View** is **On**.
Header	Sets the number of bits in the packet header. The default value is 0. Available when **Packet View** is **On**.
Trailer	Sets the number of bits in the packet trailer. The default value is 0. Available when **Packet View** is **On**.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

NRZ serial bus search configuration menu (on page 444)

Parallel Bus configuration menu

Use this menu to set up and display a parallel bus waveform. Parallel bus decoding and triggering is included with the oscilloscope.

To set up the parallel bus:

- To create a new parallel bus, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double clicking on the badge. Set the **Bus Type** to **Parallel**.
- To change the settings on an existing parallel bus waveform, double-tap the **Bus** waveform badge to open the configuration menu, and make necessary changes.

Parallel bus configuration menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **Parallel** to define a parallel bus.
Clocked Data	Toggles **Yes** or **No** to use a clock signal to recover the data bits from the bus inputs.
Clock Source	Sets the source for the bus clock signal. The source can be an analog channel. Available when **Clocked Data** is set to **Yes**.
Polarity	Sets the clock signal edge to use for timing reference. Available when **Clocked Data** is set to **Yes**.
Threshold	Sets the threshold value to determine high logic value. Available when **Clocked Data** is set to **Yes**.
Define Inputs	Opens a Parallel Bus - Define Inputs configuration menu to set the signal sources and the bit order (MSB to LSB) for the bus. See Parallel Bus - Define Inputs menu (on page 238).
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.

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

Parallel Bus - Define Inputs menu

Use this menu to select the signal sources and order for the parallel bus waveform

To access the Parallel Bus - Define Inputs menu, double-tap a Parallel Bus badge to open the configuration menu, and tap the **Define Inputs** button.

Parallel Bus - Define Inputs menu fields and controls

Field or control	Description
Parallel bus definition list	Lists the signal source and thresholds of selected channels or waveforms. The MSB is at the top of the list. To add a signal to the Parallel bus definition list, tap a source button in the Sources list. The button moves from the Sources list to the bottom of the bus list. Use the arrow buttons to the right of the field to move a selected signal up or down in the list. To remove a signal from the Parallel bus (and return it to the Sources list), tap on the signal source button. To change the threshold value for individual channels, tap in a selected Threshold field and use the assigned multipurpose knob, or double-tap the field to open the keypad and enter values.
Sources	Lists all available sources to use for a parallel bus. To add a source to the Parallel bus definition list, tap a source button. The button moves from the Sources list to the bottom of the bus list.
Set All Thresholds	Sets all thresholds in the Parallel bus definition list to the specified value. Enter a value and tap **Apply** to set the values.

See also

Bus Search configuration menus (on page 375)

Parallel bus search configuration menu (on page 445)

PCIe serial bus configuration menu

To set up a PCIe serial bus:

1. To create a new PCIe 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 **PCIe**.
2. To change the settings on an existing PCIe serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

PCIe serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.

Field or control	Description
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice in Stacked mode, or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions centered vertically in a slice or on the screen.
Bus Type	Set to **PCIe**.
Speed	Sets the speed of the PCIe protocol. Available options are **2.5 Gb/s**, **5 Gb/s**, **8 Gb/s**, and **16 Gb/s**.
Link Width	Sets the link width of the PCIe protocol. Available option is **X1**.
Signal Type	Sets the signal type to **Single-Ended** or **Differential**.
Source	Select the signal source from the listed analog channels.
Threshold	Sets the threshold value for the signal.
Packet View	Turns on or off the packet view of the bus in the waveform view.
D+ Input	Select the channel source and set the threshold level for the Data+ signal. Available only when **Signal Type** is set to **Single Ended**.
D- Input	Select the channel source and set the threshold level for the Data- signal. Available only when **Signal Type** is set to **Single Ended**.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms. Available options are **Bus** and **Bus and Waveforms**
Decode Format	Sets the decode format used to display the bus information. Available options are **Hex**, **Binary**, and **Mixed Hex**.

See also

Bus Search configuration menus (on page 375)

PCIe serial bus search configuration menu (on page 445)

PSI5 serial bus configuration menu

Use the PSI5 bus menu to set up and display an PSI5 (Peripheral Sensor Interface) serial bus waveform.

To set up the PSI5 serial bus:

- To create a new PSI5 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 **PSI5**.
- To change the settings on an existing PSI5 serial bus waveform, double-tap the PSI5 **Bus** waveform badge and make necessary changes.

PSI5 serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **PSI5** to set up and display a PSI5 bus waveform.
Direction	Select the direction.
Sensor To ECU	Sets the BUS to Sensor to ECU mode.
ECU to Sensor	Sets the BUS to ECU to Sensor mode.
Source	Set the signal source from available analog channels.
Threshold	Set the threshold level to define a logic high level.
Mode	▪ Configure the speed. ▪ Standard (125 kbps) ▪ Fast (189 kbps) ▪ Slow (83.3 kbps)
Data A	Set the number of bits for data region A.
Data B	Set the number of bits for data region B.
Serial Messaging	Turns On/Off the Serial channel messaging
Frame Control	Indicates the type of frame, data content or identifies the sensor. The values are 0,1,2,3, and 4.
Status	Set the sensor status from 0 to 2 bits.

Field or control	Description
Sync Mode	▪ Tooth Gap ▪ Pulse Width Only available when **Direction = ECU to Sensor** .
Bit Period	Only available when **Direction = ECU to Sensor** and **Sync Mode = Tooth Gap**.
Data Format	▪ Nibble ▪ Byte Only available when **Direction = ECU to Sensor** .
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

PSI5 serial bus search configuration menu (on page 449)

RS232 serial bus menu

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

To set up a RS232 serial bus:

1. To create a new RS232 bus waveform, tap the **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**.
2. To change the settings on an existing RS232 serial bus waveform, double-tap the RS232 **Bus** waveform badge to open the configuration menu and make necessary changes.

RS232 serial bus menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.

Field or control	Description
Bus Type	Set to **RS232**.
Data Inputs	Set the data input as **One** or **Two**.
Bit Rate	Sets the data bit rate. To enter a custom bit rate, select Custom and enter the custom bit rate in the **Custom Rate** input box.
Data Bits	Set the number of bits that define a data packet for your RS232 bus.
Source	Set the signal source from available analog channels.
Threshold	Set the threshold level to define a logic high level.
Polarity	Select the polarity to match the RS232 bus being acquired. Use Normal polarity for RS-232 signals, and Inverted polarity for RS-422, RS-485, and UART buses.
Parity	Set the parity to match the RS232 bus being acquired.
Packet View	Set to **On** to show decoded packet level information on the bus waveform.
End of packet	Select the appropriate end of packet value to match the RS232 bus being acquired. Available when **Packet View** is **On**.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and ASCII.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

RS-232 serial bus search configuration menu (on page 453)

SDLC serial bus configuration menu

Use the SDLC bus menu to set up and display a SDLC (Serial Data Link Control) serial bus waveform.

To create a new SDLC bus waveform:

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

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

SDLC serial bus menu fields and controls

Field or control	Description
Display	Turns On or Off. Displays the decode bus on the waveform view, when the **Display** is set to **On**.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **SDLC** to set up and display a **SDLC** bus waveform.
Modulo	Turn Modulo to **8** for 8 bit control field or Modulo to **128** for 16 bit control field in SDLC transmission frame.
Encoding	Turns **Discrete Transmission** or **Invert On Zero** SDLC line encoding for Non Return to Zero (NRZ) and Non return to Zero Invert (NRZI) standards.
Data Input	Sets the source, threshold and bitrate for the data input signal.
Source	Select the channel source for the signal from a differential probe.
Threshold	Sets the threshold level to define a logic high level. Sets the threshold value for the signal source.
Bit Rate	Sets the data bit rate. To enter a data bit rate, double-tap and enter the value in the input box.
Display Format	Sets the waveform view to show the decoded bus information.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex, Binary, and mixed Hex.

SENT serial bus configuration menu

Use the SENT bus menu to set up and display a SENT (Single Edge Nibble Transmission) serial bus waveform.

SENT serial bus menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus in the Waveform view.

Field or control	Description
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using virtual keyboard, or tap the field and enter text from an attached keyboard.
Bus Type	Set to **SENT** to set up and display a SENT serial bus waveform.
Source	Select the signal source from the listed analog channels.
Threshold	Sets the high/low logic transition level.
Polarity	
Clock Tick	Sets the time period of the clock tick. The valid range is from 1 µs to 300 µs. The default is 3 µs.
Tick Tolerance	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%.
Fast Data Channels	Sets the number of fast data channels. The default is **2**.
Data Nibbles	Sets the number of data nibbles to detect in the serial signal (**3**, **4**, or **6**). Available when **Fast Data Channels** is **1**.
Channel Widths (C1/C2)	Sets the number of bits per channel when using two fast data channels (**12/12**, **14/10**, or **16/8**). Available when **Fast Data Channels** is **2**.
Pause Pulse	Sets the instrument to detect a Pause pulse in the serial data. The default value is .
Slow Channel	Sets the slow channel characteristics. Tap and select from the available list of slow channel types. The default value is **None**.
Display Format	Sets the display format for waveforms as **Bus** or **Bus and Waveform**. **Bus** sets the waveform view to display just the decoded bus information. **Bus and Waveform** sets the waveform view to display both the decoded bus and the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**,

SMBus serial bus configuration menu

Use the SMBus menu to set up and display a SMBus serial bus waveform.

To set up the SMBus serial bus menu:

- To create a new SMBus 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 **SMBus**.
- To change the settings on an existing SMBus serial bus waveform, double-tap the SMBus **Bus** waveform badge and make necessary changes in the configuration menu.

SMBus serial bus menu fields and controls

Field or control	Description
Display	Enables or disables displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (Stacked mode), or center screen (Overlay mode). The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **SMBus**.
PEC Byte	Set PEC Byte. By default PEC Byte is false.
SMBCLK Input	Set the source and threshold level for the SMBus input clock signal.
SMBDAT Input	Set the source and threshold level for the SMBus input data signal.
Display Format	Set the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Set the decode format used to display the bus information. Available formats are Hex, Mixed Hex, and Binary.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

SMBus serial bus search configuration menu (on page 459)

SpaceWire serial bus configuration menu

Use the SpaceWire bus menu to set up and display an SpaceWire serial bus waveform.

To set up a SpaceWire serial bus:

1. To add a new SpaceWire bus waveform, tap the **Add New Bus** button on the Settings bar. Open the bus configuration menu by double-tapping the badge. Set the **Bus Type** to **SpaceWire**.
2. To change the settings on an existing SpaceWire serial bus waveform, double-tap the **SpaceWire** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

You can configure search for SpaceWire (on page 462) and define conditions to search for and mark on an SpaceWire bus waveform.

SpaceWire serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to Bus type to **SpaceWire**.
Strobe Input	Sets the source and threshold level for the signal.
Data Input	Sets the source and threshold level for the signal.
Sync	Sets the synchronization. Available options are Data, Auto, NULL, and Custom.
Data	Set the data. Available when **Sync** = **Data**. Use Configure pattern editor (on page 396) to customize the data.
Count	Displays the count. Available when **Sync** = **NULL**.
Custom	Set the custom value. Available when **Sync** = **Custom**. Use Configure pattern editor (on page 396) to customize the custom value.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are Hex and Binary.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

SpaceWire serial bus search configuration menu (on page 462)

SPI serial bus configuration menu

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

To set up the SPI serial bus:

1. To create a new SPI bus waveform, tap the **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**.
2. To change the settings on an existing SPI serial bus waveform, double-tap the **Bus** waveform badge to open the configuration menu, and make necessary changes.

SPI serial bus menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen.
Bus Type	Set to **SPI**.
Data Inputs	Select the number of inputs as **One** or **Two**. One enables to configure one data input. Two enables to configure MOSI and MISO.
Framing	Set to Slave Select (SS) or Idle framing modes.
SCLK Input	Select the channel source and threshold level for the Serial Clock signal (output from master). Set the Polarity to rising or falling edge of the clock signal used by the master device to start transferring bits. Available when **Framing** is **SS** and **Data Inputs** is **One**, and **Framing** is **Idle**.
SS Input	Select the channel source and threshold level for the Slave Select signal to start communications with the slave device. Set the Polarity to use Active High or Active Low logic for the SS signal. Available when **Framing** is **SS**.

Field or control	Description
Data Input	Select the channel source and threshold level for the data input. Set the Polarity to use Active High or Active Low for the SS signal. Available when **Data Inputs** is **One**.
MOSI Input	Select the channel source and threshold level for the MOSI (Master In Slave Input) input. Set the Polarity to use Active High or Active Low logic for the MOSI signal. Available when **Data Inputs** = **Two**, and **Framing** = **SS/Idle**.
MISO Input	Select the channel source and threshold level for the MISO (Master In Slave Output) input. Set the Polarity to use Active High or Active Low logic for the MISO signal. Available when **Framing** is **SS/Idle** and **Data Inputs** is **Two**.
Word Size	Enter the word size, in bits. Minimum is 4, maximum is 32, and default is 8.
Bit Order	Set to most significant bit **MS First** or least significant bit **LS First**.
Idle Time (Framing =)	Set the idle frame time. Available when **Framing** is **Idle**.
Display Format	Sets the waveform view to show just the decoded bus information, or the decoded bus plus the source signal waveforms. This depends on data input. Available only when **Data Input** is **One**.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex** and **Binary**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

SPI serial bus search configuration menu (on page 460), SPI serial bus search configuration menu (on page 247)

SPMI serial bus configuration menu

Use the SPMI bus menu to set up and display an SPMI (System Power Management Interface) serial bus waveform.

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
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set to **SPMI** to set up and display a SPMI bus waveform.
SCLK Input	Sets the signal **Source** and the high/low logic transition **Threshold** level for the **SCLK** input signal.
SDATA Input	Sets the signal **Source** and the high/low logic transition **Threshold** level for the **SDATA** input signal.
Display	**Bus** sets the waveform view to show just the decoded bus information. **Bus and Waveforms** 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.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

SPMI serial bus search configuration menu (on page 461)

SVID serial bus configuration menu

Use the SVID bus menu to set up and display an SVID (Serial VID (clock, data, alert)) serial bus waveform.

To set up the SVID serial bus:

- To create a new SVID 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 **SVID**.
- To change the settings on an existing SVID serial bus waveform, double-tap the SVID **Bus** waveform badge and make necessary changes.

SVID serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically in a slice or on the screen).
Bus Type	Set the **SVID** to set up and display a SVID bus waveform.
Version	Sets the version to **Non VR14** or **VR14**.
Clock Input	Set the source and threshold level for the clock input signal.
Data Input	Sets the source and threshold level for the data input signal.
Alert Input	Sets the source and threshold level for the alert input signal.
Threshold	Set the threshold level to define a logic high level. Sets the threshold value for the signal source.
Display Format	Sets the waveform view to show the decoded bus information, or the decoded bus and the source signal waveforms.
Decode Format	Sets the decode format used to display the bus information. Formats are **Hex**, **Binary**, and **Mixed Hex**.

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

SVID serial bus search configuration menu (on page 464)

USB serial bus configuration menu

Use the USB bus menu to set up and display an USB (Universal Serial Bus) waveform.

To set up a USB serial bus:

1. 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**.
2. To change the settings on an existing USB serial bus waveform, double-tap the **Bus** waveform badge on the Settings bar to open the configuration menu, and make necessary changes.

USB serial bus menu fields and controls

Field or control	Description
Display	Turns on or off displaying the bus on the Waveform view.
Label	Enter a label for the bus. The default label is the selected bus type. To enter label text, double-tap the field and enter label using the virtual keyboard, or tap the field and enter text from an attached keyboard.
Position	Sets the vertical position of the bus waveform. The default position is vertically centered in a slice (in Stacked mode), or center screen in Overlay mode. The unit of position is screen divisions.
Set to 0	Sets the vertical position to 0 divisions (centered vertically) in a slice or on the screen.
Bus Type	Set to **USB**.
Speed	Set the speed to match the USB bus you are acquiring. The speed options depend on the installed license and may not all be visible at the same time. Depending on the license, the Speed setting may offer one or more of the following options: Available options are USB 1.0 (1.5 Mbps), USB 1.1 (12 Mbps), USB 2.0 (480 Mbps), USB 3.0 (5 Gbps), USB 3.1 Gen 1 (5 Gbps), USB 3.2 Gen 1 (5 Gbps), USB 3.1 Gen 2 (10 Gbps), USB 3.2 Gen 2 (10 Gbps), USB 4.0 Gen 2 (10 Gbps), and USB 4.0 Gen 3 (20 Gbps). Selecting USB 2.0 (480 Mbps) sets the **Signal Type** to **Differential** only.

Field or control	Description
Signal Type	Set to match the USB signal you are acquiring (Single Ended or Differential). Use a differential probe to acquire the differential USB signal. Both Single Ended and Differential can be used to measure USB 1.0 (1.5 Mbps), USB 1.1 (12 Mbps), USB 3.0 (5 Gbps), USB 3.1 Gen 1 (5 Gbps), USB 3.1 Gen 2 (10 Gbps), USB 4.0 Gen 2 (10 Gbps), and USB 4.0 Gen 3 (20 Gbps) speed USB signals. Selecting USB 2.0 (480 Mbps) sets the **Signal Type** to **Differential** only.
Lanes	Sets the lane as **Single** or **Dual**. Available when **Speed** is **USB 3.2 Gen 1 (5 Gbps)** or **USB 3.2 Gen 2 (10 Gbps)**.
Packet View	Toggle **On** or **Off** the packet decode of the bus in the waveform view. Packet view is available when **Speed** is set to **USB 3.0 (5 Gbps)**, **USB 3.1 Gen 1 (5 Gbps)**, **USB 3.2 Gen 1 (5 Gbps)**, **USB 3.1 Gen 2 (10 Gbps)**, **USB 3.2 Gen 2 (10 Gbps), USB 4.0 Gen 2 (10 Gbps), and USB 4.0 Gen 3 (20 Gbps)**.
Use RS-FEC	Select the **Use RS-FEC** box to enable the error correction. Available when **Speed** is **USB 4.0 Gen 2 (10 Gbps) and USB 4.0 Gen 3 (20 Gbps).**
Source	Select the channel source for the signal from a differential probe. Source control is available when **Signal Type** is set to **Differential**.
Threshold	Sets the threshold level for the differential signal. Threshold is available when **Speed** is set to **USB 3.0 (5 Gbps)**, **USB 3.1 Gen 1 (5 Gbps)**, **USB 3.2 Gen 1 (5 Gbps)**, **USB 3.1 Gen 2 (10 Gbps)**, **USB 3.2 Gen 2 (10 Gbps), USB 4.0 Gen 2 (10 Gbps), and USB 4.0 Gen 3 (20 Gbps)**.
Lane 0	Sets the channel **Source** and the **Threshold** level for the lane 0. Available when **Speed** is **USB 3.2 Gen 1 (5 Gbps)**or **USB 3.2 Gen 2 (10 Gbps)** and **Lanes** is **Dual**.
Lane 1	Sets the channel **Source** and the **Threshold** level for the lane 1. Available when **Speed** is **USB 3.2 Gen 1 (5 Gbps)**or **USB 3.2 Gen 2 (10 Gbps)** and **Lanes** is **Dual**.
High Threshold	Sets the high threshold level for the differential signal. High Threshold control is available when **Speed** is set to **USB 1.0 (1.5 Mbps)**, **USB 1.1 (12 Mbps)**, or **USB 2.0 (480 Mbps)**.
Low Threshold	Sets the low threshold level for the differential signal. Low Threshold control is available when **Speed** is set to **USB 1.0 (1.5 Mbps)**, **USB 1.1 (12 Mbps)**, or **USB 2.0 (480 Mbps)**.
D+ Input	Select the channel source and set the threshold level for the Data+ signal. Available only when **Signal Type** is set to **Single Ended**.

Field or control	Description
D- Input	Select the channel source and set the threshold level for the Data- signal. Available only when **Signal Type** is set to **Single Ended**.
Display Format	Sets the waveform view to show just the decoded bus information.
Decode Format	Sets the decode format used to display the bus information. Available options are: ▪ For USB 1.0/1.1/2.0 speed signal: **Hex**, **Binary**, **Mixed ASCII**, and **Mixed Hex** ▪ For USB 3.0/3.1 Gen 1/3.2 Gen 1/3.1 Gen 2/3.2 Gen 2, USB 4.0 Gen 2/Gen 3 speed signal: **Hex**, **Binary**, and **Mixed Hex.**

Other bus types

Other serial bus types 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 (when supported) to the **Trigger** menu. See the Bus configuration menu (on page 204) for links to all serial bus configuration menus.

See also

Bus Search configuration menus (on page 375)

USB serial bus search configuration menu (on page 465)

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
Clear	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.
Enter	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.
Cancel	Closes the logic keypad without saving entered values.

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.

Cursor configuration menu fields and controls

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

Field or control	Description
Display	Tap to toggle the cursor display **On** or **Off**.
Readout Location	Tap to set where to display cursor readouts. **Graticule** displays the cursor readouts on the screen cursor bars (default method). You can move the readouts on cursors by touching and dragging them along the cursor bar. **Badge** removes the screen cursor readouts and displays the cursor information in a **Cursors** badge in the Results bar. The **Cursors** badge content changes as you change the cursor type.
Cursor Type	Tap to select the cursor type from the drop-down list. **Waveform** cursors measure vertical amplitude and horizontal time parameters simultaneously at the point the cursor intersects a waveform. The cursor intersect point tracks waveform amplitude changes. **V Bars** are vertical cursors that measure horizontal parameters (typically time). They are not associated with the waveform, but show the time position of the cursor in the waveform record. **H Bars** are horizontal cursors that measure amplitude (typically in volts or amperes). They are not associated with the waveform, but show the amplitude position of the cursor. **V&H Bars** cursors measure vertical and horizontal parameters simultaneously. They are not associated with the waveform, but show the time and amplitude position of the cursors. **Custom** cursors measures waveform values using multiple user-defined vertical and horizontal cursors. Displays cursor position information and delta measurements between cursors. Readout information can be displayed on the graticule or in a badge. Available only for time-domain waveforms and Time Trend plots.
Source Waveform	Tap to select the source waveform (the waveform on which to display the cursors). **Same** places both cursors on the same waveform. **Split** allows each cursor to be on a different waveform. Available when **Cursor Type** is **Waveform**. If **Source** is set to **All** then the **Source Waveform** is unavailable.

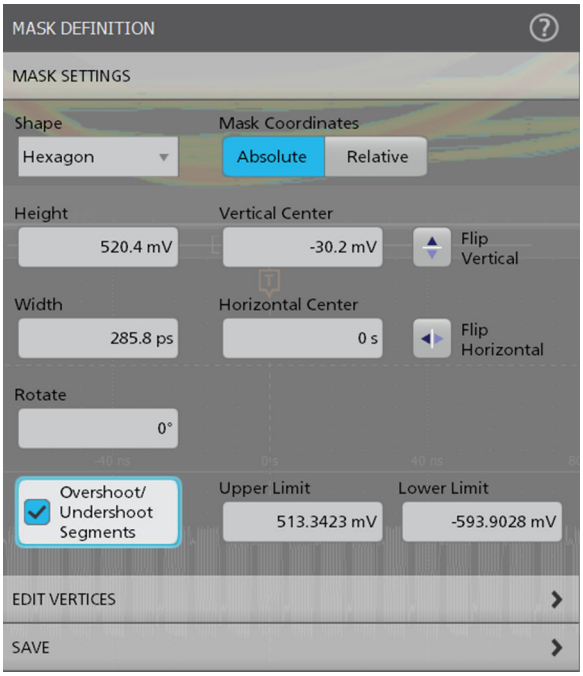
Field or control	Description
Readout Type	Tap to select the type of readout to display for a XY plot. This control is only available when the **Cursor Type** is set to Waveform or **V&H Bars**. **Rectangular** displays the position values and the difference (delta) between cursors. **Polar** displays the polar information for each cursor. The top value is the magnitude of the vector drawn from the origin of the plot to the location of the cursor. The bottom value is the angle of the vector drawn from the origin of the plot to the location of the cursor.
Source	Tap to select the source waveform from the drop-down list. **Selected Waveform** automatically moves the waveform cursors to the selected source or select a specific source from the drop-down list. Available when **Source Waveform** is **Same**. **All** is available in **Source** drop-down list only when the **Cursor Type** is set to **Waveform**. **All** waveforms includes time domain based math waveforms, reference waveforms, trend waveforms, etc.
Cursor A Source, Cursor B Source	Tap to select the waveform sources for Cursor A and B. Available when **Source Waveform** is **Split**.
Cursor A X-Position	Tap to set a specific x-axis position for Cursor A using the multipurpose knob, or double-tap to set the position using the keypad.
Cursor B X-Position	Tap to set a specific x-axis position for Cursor B using the multipurpose knob, or double-tap to set the X-Position using the keypad.
Cursor A Y-Position	Tap to set a specific Y-axis position for Cursor A using the multipurpose knob, or double-tap to set the position using the keypad. Available when **Cursor Type** is **H Bars** or **V&H Bars**.
Cursor B Y-Position	Tap to set a specific Y-axis position for Cursor B using the multipurpose knob, or double-tap to set the X-Position using the keypad. Available when **Cursor Type** is **H Bars** or **V&H Bars**.
Cursor Mode	Tap to select the cursor mode. **Independent** mode sets multipurpose knobs A and B to move each cursor separately. **Linked** mode sets multipurpose knob A to move both cursors at the same time. Knob B will still move cursor B independently of knob A. Available when **Cursor Type** is **Waveform**, **V Bars**, **H Bars** or **V&H Bars**.
Adjust Setting For	Tap to select **Vertical** Cursors or **Horizontal** Cursors. By default **Vertical** Cursors is enabled. When **Vertical** Cursors is selected, the menu displays controls for configuring vertical cursors. When **Horizontal** Cursors is selected, the menu displays controls for configuring horizontal cursors. Available when **Cursor Type** is **Custom**.

Field or control	Description
On Horizontal Scale Change, Maintain	Tap to select **Screen Position** or **X-Position** to change the scale position. By default **Screen Position** is enabled. Select Screen Position to maintain the cursor screen position. Select X-Position to maintain the cursor X-position. Available when **Cursor Type** is **Custom**.
Add Cursor	Adds a new cursor row to the end of the cursor table. Available when **Cursor Type** is **Custom**.
Delete Cursor	Deletes the selected cursor row from the cursor table. Available when **Cursor Type** is **Custom**.

Eye Mask Definition configuration menu

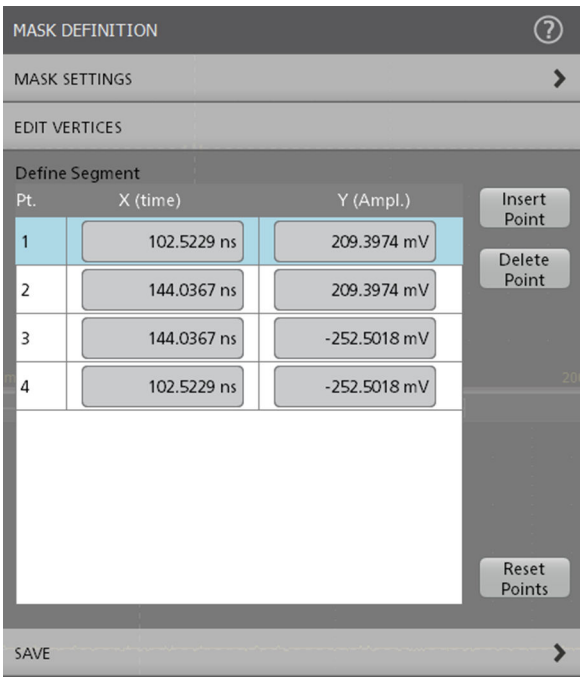
Use the Mask Definition menu to edit mask segment parameters. Double-tapping the central mask segment opens the Mask Definition configuration menu. All the settings here except for Overshoot/Undershoot Segments, Upper Limit and Lower Limit are specified for the central segment.

To create a mask segment, see Create a Mask on waveforms (on page 114).



Mask Settings panel fields and controls

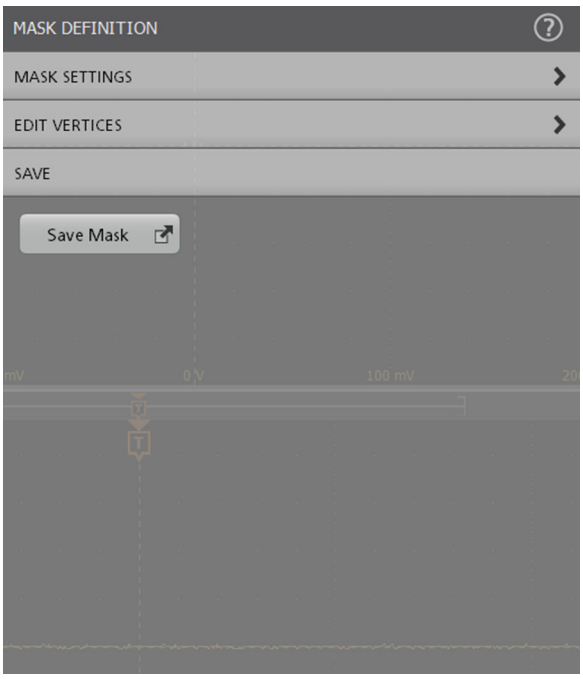
Field or control	Description
Shape	Displays the current shape of the central segment and lets you change it to a different shape. When you change a segment's shape, the system calculates the smallest rectangle that contains all vertices of the central segment and then uses a best-fit approach to generate the specified shape. If you modify a segment that originally used one of the default shapes and your changes cause it to no longer meet that shape's criteria, the segment appears as a Custom shape in the menu.
Height	Sets the central segment height between the topmost vertex and the bottommost vertex.
Vertical Center	Sets the central segment vertical center as the point halfway between the topmost vertex and bottommost vertex.
Flip Vertical	Flips the central segment vertically around its Vertical Center value
Width	Sets the central segment width between the leftmost vertex and the rightmost vertex.
Horizontal Center	Sets the central segment horizontal center, in specified units, as the point halfway between the leftmost vertex and rightmost vertex.
Flip Horizontal	Flips the segment horizontally around its Horizontal Center value.
Rotate	Rotates the segment in units of degrees, from 0° to 360°. The rotation is an absolute angle measurement referenced from 0°, where 0° is the position of the segment when it was first created. For example, if you rotate the segment 40°, and then rotate it again with 20°, the resulting segment rotation is 20°. The segment height and width are relative to the current segment orientation, and automatically change places as needed when the segment is rotated.
Overshoot/Undershoot Segments	Adds or removes the overshoot and undershoot segments.
Upper Limit	The Y coordinate of the bottom of the overshoot segment.
Lower Limit	The Y coordinate of the top of the undershoot segment.



Edit Vertices panel fields and controls

Field or control	Description
Define Segment	A table that lists the X (time) and Y (amplitude) values for each vertex (point) of the segment. Use the scroll bar to show points if there are more than nine points in the segment. Selecting a row in the table highlights the associated vertex on the segment. Use the Multipurpose knobs to change the value of the X or Y settings, or double-tap a setting and enter the value directly.
Insert Point	Inserts a new row above the selected row and creates a new vertex on the segment shape. The new vertex is halfway between the vertices defined in the prior row and the following row in the table.
Delete Point	Deletes the currently selected point, keeps the row selected, and moves all rows below it up one row. The **Delete Point** button is not available when a triangular segment is selected.
Reset Points	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.

Redo and Undo are available for most Edit Vertices panel controls.



Save panel fields and controls

Field or control	Description
Save Mask	Saves the mask in the desired location with the given file name.

Right click menu functions associated with the central mask segment

Use right-click menu associated with the central mask segment provides options to modify the segment.

Field or control	Description
Triangle	Sets the shape of the mask to a triangle.
Rectangle	Sets the shape of the mask to a rectangle.
Trapezoid	Sets the shape of the mask to a trapezoid.
Hexagon	Sets the shape of the mask to a hexagon.
Configure Mask	Opens the Mask Definition configuration menu for the mask.
Delete Mask	Deletes the mask on the eye diagram.

Date and Time configuration menu

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

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

Date and Time configuration menu fields and controls

Field or control	Description
Display	Tap to toggle display of the date and time On or Off. When turned off, double-tap on the blank area below the Run/Stop button, in the lower-right corner of the oscilloscope display, to open the configuration menu and set display to On.
Time Format	Tap to select either 12 Hour or 24 Hour time format.
Time Zone	Tap and select the desired time zone from the drop-down list.
Automatically adjust clock for Daylight Saving Time	Sets the oscilloscope to automatically change the time to account for Daylight Saving Time changes.

NOTE: You cannot set a specific time; the time is preset at the factory.

Font Color menu (Text Settings configuration)

Use this menu to change the label color. Touch and hold on callout 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.

Horizontal configuration menu

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

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

Horizontal configuration menu fields and controls

Field or control	Description
Horizontal Mode	Tap to select either Automatic or Manual horizontal mode. In **Automatic** mode you can set the **Minimum Sample Rate** and **Horizontal Scale**. In **Manual** mode you can set the Sample Rate and Record Length. Horizontal Scale is a dependent variable calculated from the sample rate and record length.
FastAcq	Tap to toggle **FastAcq** mode **On** or **Off**. **FastAcq** provides high-speed waveform capture. It is helpful in finding elusive signal anomalies. Fast Acq mode reduces the dead time between waveform acquisitions, enabling the capture and display of transient events such as glitches and runt pulses. Fast Acq mode can also display waveform phenomena at an intensity that reflects their rate of occurrence. If the waveforms are overlapping while saving the FastAcq data, use Stacked Display mode without turning on the zoom. During waveform save if FastAcq is on, the pixmap data is saved into the file (not sampled waveform data or vector) but the saved data cannot be recalled. The FastAcq supports only .csv file format.

Field or control	Description
FastAcq Palette	Tap and select the desired FastAcq Palette from the drop-down list. **Temperature** uses color-grading to indicate frequency of occurrence, with hot colors like red/yellow indicating frequently occurring waveform data points, and colder colors like blue/green indicating rarely occurring data points. **Spectral** uses color-grading to indicate frequency of occurrence with, colder colors like blue indicating frequently occurring waveform data points, and hot colors like red indicating rarely occurring data points. **Normal** uses the default channel color (like yellow for channel one) along with gray-scale to indicate frequency of occurrence where frequently occurring events are bright. **Inverted** uses the default channel color (like yellow for channel one) along with gray-scale, but inverts color intensity such that events that occur less frequently are brighter. Available when **FastAcq** is **On**.
Minimum Sample Rate	Sets the minimum sample rate for acquisitions. Changing horizontal scale will not reduce the sample rate below this value. This can result in partial waveform records at lower frequencies, but at the same time would provide more sample points for a few cycles of the signal. This setting can be overridden if **Allow Horizontal Scale to Override Min Sample Rate** is selected.
Allow Horizontal Scale to Override Min Sample Rate	Select to set the oscilloscope to automatically change the sample rate to acquire a full waveform record. Changes to the sample rate are shown in the Horizontal badge readout, not in the Minimum Sample rate field of the menu.
Maximum Record Length	Tap and select the Maximum Record Length from the drop-down list. Record length is dependent on the Scale setting. If changing the scale causes the record length to exceed the maximum record length Limit, the sample rate is decreased to the next available setting to keep the record length below the set maximum. Available when **Horizontal Mode** is **Automatic** and **FastAcq** is **On**.
Allow Horizontal Scale to Override Max Record Length	Select to set the oscilloscope to automatically change the record length to acquire a full waveform record. Changes to the sample rate are shown in the Horizontal badge readout, not in the Minimum Sample rate field of the menu. Available when **Horizontal Mode** is **Automatic** and **FastAcq** is **On**.
Horizontal Scale	Tap to set the Horizontal Scale using the assigned multipurpose knob, double-tap to set the scale using the virtual keypad, or tap the up and down arrows. You can also use the front-panel Horizontal Scale knob to change this value. The horizontal scale determines the size of the acquisition window relative to the waveform. You can scale the window to contain a single waveform edge, a single cycle, several cycles, or thousands of cycles.
Delay	Delay positions the trigger event to a specified time relative to the center of the waveform record. Use delay to focus on events that occur before (pre-trigger) or after the trigger point (post-trigger).

Field or control	Description
Position	Tap to set the trigger Position using the assigned multipurpose knob or double-tap to set the Position using the virtual keypad. When horizontal Delay is on, the time from the trigger point to the horizontal reference (center of waveform record) is the horizontal delay. The horizontal position determines the number of pre-trigger and post-trigger samples in the waveform record. When horizontal delay is off, the trigger point and the horizontal reference are at the same time in the middle of the waveform record.
Set to 0 s	Tap to set the delay position to 0 s (center of the waveform record. Available only when **Delay** is **On**.
Set to 10%	Tap to set the trigger delay to 10% of the waveform record. Available only when **Delay** is **Off**.
Sample Rate Changes Affect	Tap to allow changes to the Sample Rate to affect either the Horizontal Scale or the Record Length. Available only when **Horizontal Mode** is **Manual**.
Sample Rate	Tap to set the Sample Rate using the assigned multipurpose knob, double-tap to set the rate using the virtual keypad, or tap the up and down arrows. This keeps the oscilloscope at the specified sample rate regardless of the horizontal or record length settings. This limits the available horizontal scale settings to those that can be used with the specified sample rate. Available only when **Horizontal Mode** is **Manual**.
Record Length	Tap to set the Record Length using the assigned multipurpose knob, double-tap to set the length using the virtual keypad, or tap the up and down arrows. Available only when **Horizontal Mode** is **Manual**.

Jitter measurement configuration menu overview

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

To open the Jitter measurement configuration menu for a measurement, double-tap a Jitter 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 Jitter measurement name fields are listed in the following table.

Jitter Measurement configuration menu fields, controls, and panels

Field, control, or panel	Description
Measurement Statistics (Measurement name panel)	A list of measurement statistics related to the measurement. You can add these to a measurement badge by selecting the **Show Statistics in Badge** control.

Field, control, or panel	Description
Show Statistics in Badge (Measurement name panel)	Adds the listed statistical measurement readouts to the measurement badge readout.
Plots (Measurement name panel)	Opens Plot views of the measurement. Available plots depend on the measurement. Plot types include Bathtub, TIE Spectrum, Time trend plot, TIE Histogram, Eye Diagram plot, and Composite Jitter Histogram. To add a plot to the screen, tap the Plot button. See Add Plot configuration menu (on page 344).
Configure panel	Sets the source, label text, and other fields that are specific to each measurement type. See Configure panel (Jitter measurement configuration menu) (on page 264).
Reference Levels panel	Sets the reference levels and units used to take measurements, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values. See Reference Levels panel (Measurement configuration menu) (on page 303).
Clock Recovery panel	Sets the clock recovery settings for some jitter measurements. See Clock Recovery panel (Measurement configuration menu) (on page 304).
Gating panel	Sets the measurement region (gate) used to take measurements. Select the scope of the gate setting (global or local), and the type of gating to use. See Gating panel (Measurement configuration menu) (on page 309).
Filter/Limit Results panel	Sets the scope of the filtering setting (global or local), high and low pass filter settings, the range of measurement result limits, and the limit measurement population size. See Filter/Limit Results panel (Measurement Settings menu) (on page 310).
Pass/Fail Testing panel	Sets the conditions for pass/fail testing for measurements and sets actions to take when a measurement fails. See Pass/Fail Testing panel (Measurement Settings menu) (on page 311).

Jitter Measurement Name panel (Measurement configuration menu)

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

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

Field or control	Description
Measurement Statistics	Shows a list of measurement statistics. You can add these statistics to a measurement badge by selecting the **Show Statistics in Badge** control.
Show Statistics in Badge	Adds the listed statistical measurement readouts to the measurement badge.
Plots	Adds a Plot view of the measurement value to the screen. Available plots depend on the measurement. Plot types include Bathtub, TIE Spectrum trend plot, TIE Histogram, Eye Diagram plot, and Composite Jitter Histogram. Trend adds the trend plot to the Waveform view.

See also

Configure panel (Jitter measurement configuration menu) (on page 264)

Reference Levels panel (Measurement configuration menu) (on page 303)

Clock Recovery panel (Measurement configuration menu) (on page 304)

Gating panel (Measurement configuration menu) (on page 309)

Filter/Limit Results panel (Measurement Settings menu) (on page 310)

Pass/Fail Testing panel (Measurement Settings menu) (on page 311)

Configure panel (Jitter measurement configuration menu)

Use the configure panel to add a custom name (label) for the measurement and other parameters.

To open the Configure panel for a Jitter measurement:

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

Amplitude Measurements panel

Bit Low, Bit High, and Bit Amplitude measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Bit Type	Sets the waveform bit types as **Trans**, **Non-Trans**, or **All**. Trans bit type does analysis only on transition bits. A transition bit is a bit that is changing from low to high or high to low. Non-trans bit type does analysis only on non-transition bits. A non-transition bit is a bit that is not changing state. All bit type does analysis using both transition and non-transition bits.
Measure the Center % of Bit	Set the center % of bit. The default value is 1%.

Field or control	Description
Method	Set the method as **Mean** or **Mode**.

DC Common Mode measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Source 2	Select the input signal channel from the Source 2 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

AC Common Mode (PK-PK) measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Source 2	Select the input signal channel from the Source 2 fields. Source 2 field enables when **Measurement Type** is **Double**.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Measurement Type	Set the measurement type as **Single** or **Double**.
>30 kHz High Pass Filter	Enables or disables the high pass filter.

Differential Crossover measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Source 2	Select the input signal channel from the Source 2 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Main Edge	Set the edge type.

T/nT measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

Eye Measurements panel

Eye Height measurement: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Source 2	Select the input signal channel from the Source 2 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Bit Type	Sets the waveform bit types as **Trans**, **Non-Trans**, or **All**. Trans bit type does eye analysis only on transition bits. A transition bit is a bit that is changing from low to high or high to low. Non-trans bit type does eye analysis only on non-transition bits. A non-transition bit is a bit that is not changing state. All bit type does eye analysis using both transition and non-transition bits.
Analysis Method	Set the analysis method as **Parametric** or **Histogram** (Non-parametric) for the measurement.
Set Levels In	Sets the levels as **%** or **Absolute**. It defines the exact time in percentage terms or absolute values.
Measure at % of the Unit Interval	Set the horizontal position where the measurement is taken as a percentage of the unit interval.Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Set Levels In** is **%**.
Measure at 'X' seconds	Sets the horizontal position where the measurement is taken in terms of seconds. Available when **Set Levels In** is **Absolute**.
Width	Sets the window around the set level to search for the maximum eye width. Available when **Analysis Method** is **Histogram**.

Eye Width measurement: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Source 2	Select the input signal channel from the Source 2 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Analysis Method	Set the analysis method as **Parametric** or **Histogram** (Non-parametric) for the measurement.
Set Levels In	Sets the levels as **%** or **Absolute**. It defines the exact time/voltage point in percentage terms or absolute values. Available when **Analysis Method** is **Histogram**.
Measure at % of the Unit Amplitude	Sets the horizontal position where the measurement is taken, as a percentage of the unit amplitude. Available when **Analysis Method** is **Histogram** and **Set Levels In** is **%**.
Measure at 'X' voltage	Sets the vertical position where the measurement is taken in terms of volts. Available when **Analysis Method** is **Histogram** and **Set Levels In** is **Absolute**.
Width	Sets the window around the set level to search for the maximum eye width. Available when **Analysis Method** is **Histogram**.

Height@BER measurement: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Bit Type	Sets the waveform bit types as **Trans**, **Non-Trans**, or **All**. Available only when Jitter Analysis Method is set to **Jitter Only**.
Measurement Range **Start**, **Finish**	Sets the measurement range. The default value is 50%. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available only when Jitter Analysis Method is set to **Jitter Only.**
Number of Bins	Set the number of bins. Defines the resolution by the number of bins into which Span is divided: 25, 50, 100, 250, 500, 2000, or Maximum. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available only when Jitter Analysis Method is set to **Jitter Only**.

Field or control	Description
Target BER	Sets the Bit Error Rate, thereby setting the statistical level at which total jitter and eye opening are reported. Default value is 1e-12. The minimum value is 1e-18 and maximum value is 1e-2. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.
Signal Type	Sets the signal type to **Clock**, **Data**, or **Auto**. Available only when Jitter Analysis Method is set to **Jitter + Noise.**
Pattern Detection	Sets the pattern detection method to **Auto** or **Manual**. ▪ **Auto** automatically detects the data pattern on the first acquisition after a configuration change and configures the Pattern Type and related controls for the selected record length. ▪ **Manual** requires you to manually set the Pattern Type and related controls. Available only when Jitter Analysis Method is set to **Jitter + Noise.**
Pattern Type	Sets the pattern type to **Repeating** or **Arbitrary**. Select **Repeating** when the data signal contains a repeating pattern of N bits. Select **Arbitrary** when the data pattern is non-repeating or unknown. Available when Pattern Detection is set to Auto. Available only when Jitter Analysis Method is set to **Jitter + Noise.**
Pattern Length	Specifies the pattern length. The default value is **2 UI**. When Pattern Detection is set to **Auto**, this field displays the detected pattern length. When Pattern Detection is set to **Manual**, set this value to match the actual pattern length. If the specified value does not match the detected pattern length, the process stops and a warning is logged. Double-tap the field to edit the value using a connected keyboard or the virtual keyboard. Available when Pattern Type is **Repeating**. Available only when Jitter Analysis Method is set to **Jitter + Noise.**
Window Length	Specifies the window length. The default value is **10 UI**. Determines the number of unit intervals over which pattern correlation effects are analyzed. Set the window length high enough to allow the impulse response of the serial data transmitter and channel to settle. Double-tap the field to edit the value using a connected keyboard or the virtual keyboard. Available when Pattern Type is **Arbitrary**. Available only when Jitter Analysis Method is set to **Jitter + Noise**.

Field or control	Description
Set Levels In	Sets the measurement levels to **%** or **Absolute**. This setting defines the measurement position using either percentage values or absolute time values. Available only when Jitter Analysis Method under User Preferences is set to **Jitter + Noise**.
Measure at 'X' Seconds	Sets the horizontal position at which the eye measurement is made in seconds. Available when Set Levels In is set to **Absolute**. Available only when Jitter Analysis Method under User Preferences is set to **Jitter + Noise**.
Measure at % of the Unit Interval	Sets the horizontal position at which the eye measurement is made as a percentage of the unit interval. Available when Set Levels In is set to **%**. Available only when Jitter Analysis Method under User Preferences is set to **Jitter + Noise**.

Width@BER measurement: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Signal Type	Set the signal type as **Clock**, **Data**, or **Auto**.
Clock Edge	Set the edge of the clock signal. Available clock edges are rising, falling, and both. Available when **Signal Type** is **Clock** or **Auto**. Available only when Jitter Analysis Method is set to **Jitter**.
Pattern Detection	Set the pattern detection method as **Auto** or **Manual**. Auto method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. Manual method allows (and requires) that the Pattern Type and associated controls be set manually.

Field or control	Description
Pattern Type	Set the pattern type as **Repeating** or **Arbitrary**. If the data signal is repeating pattern of N bits, then Repeating pattern type should be selected. If the data signal is non-repeating pattern or unknown then Arbitrary pattern type should be selected. Available when **Pattern Detection** is **Auto**.
Pattern Length	Enter the pattern length. Default length is 2 UI. When Pattern Detection is Auto, this field shows the detected pattern length. When Pattern Detection is Manual, this control must be set to match the actual pattern length. If the manually-set pattern length is inconsistent with the detected pattern length, process stops but a warning will be logged. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Repeating**.
Window Length	Enter the window length. Default length is 10 UI. Determines the number of unit intervals over which pattern correlation effects are analyzed. The window should be set to a large enough value that the impulse response of the serial data transmitter and channel have settled. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Arbitrary**.
Target BER	Sets the Bit Error Rate, thereby setting the statistical level at which total jitter and eye opening are reported. Default value is 1e-12. The minimum value is 1e-18 and maximum value is 1e-2. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.

Eye High, Eye Low, and Q-Factor measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Bit Type	Sets the waveform bit types as **Trans**, **Non-Trans**, or **All**. Available only when Jitter Analysis Method is set to **Jitter Only**. Trans bit type does eye analysis only on transition bits. A transition bit is a bit that is changing from low to high or high to low. Non-trans bit type does eye analysis only on non-transition bits. A non-transition bit is a bit that is not changing state. All bit type does eye analysis using both transition and non-transition bits.
Measure at % of the Unit Interval	Set the horizontal position where the measurement is taken as a percentage of the unit interval. The default value is 50%.

Jitter Measurements panel

NOTE: For Jitter Measurements, Base Jitter Measurement Technique, Where N= and Clock Edge configurations are available only when Jitter Analysis Method is set to Jitter Only.

CC-Jitter measurement: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Edge	Set the edge of the signal as **First**, **Rising**, or **Falling**.
N=	Specifies number of cycles or unit interval in each N-period group. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.

DCD and DDJ measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Base Jitter Measurement Technique	Sets the Base Jitter Measurement Technique as **TIE**, **Period**, or **N-Unit Interval**. Tap **Utility > User Preference > JITTER AND EYE ANALYSIS** and select **Enable Jitter Measurement Based on N-Unit Interval** checkbox to enable this configuration. **TIE:** This involves Jitter analysis based on Time Interval Error (TIE). **N-Unit interval:** N-Unit Interval(N-UI) jitter analysis refers to analyzing various jitter components based on the intervals of N bits or N Unit Intervals. N-UI and Period jitter can be performed on clock or 1/0 data waveforms only. Because N-UI and period jitter are measured from edge to edge, no clock recovery is required. It is useful for capturing low-frequency jitter components and for better understanding of correlated jitter. **Period:** This is the same as N-Unit Interval based jitter measurement when N=2.
Signal Type	Set the signal type as **Clock**, **Data**, or **Auto**. This configuration is not available when **Base Measurement Technique** is set to other than **TIE**.
Where N =	Sets the value of N for unit inverval. Available when **Base Jitter Measurement Technique** is set to **N-Unit Interval**. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.

Field or control	Description
Pattern Detection	Set the pattern detection method as **Auto** or **Manual**. Auto method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. Manual method allows (and requires) that the Pattern Type and associated controls be set manually.
Pattern Type	Set the pattern type as **Repeating** or **Arbitrary**. If the data signal is repeating pattern of N bits, then Repeating pattern type should be selected. If the data signal is non-repeating pattern or unknown then Arbitrary pattern type should be selected. Available when **Pattern Detection** is **Auto**.
Pattern Length	Enter the pattern length. Default length is 2 UI. When Pattern Detection is Auto, this field shows the detected pattern length. When Pattern Detection is Manual, this control must be set to match the actual pattern length. If the manually-set pattern length is inconsistent with the detected pattern length, process stops but a warning will be logged. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Repeating**.
Window Length	Enter the window length. Default length is 10 UI. Determines the number of unit intervals over which pattern correlation effects are analyzed. The window should be set to a large enough value that the impulse response of the serial data transmitter and channel have settled. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Arbitrary**.

ISI measurement: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

Field or control	Description
Base Jitter Measurement Technique	Sets the Base Jitter Measurement Technique as **TIE**, **Period**, or **N-Unit Interval**. Tap **Utility > User Preference > JITTER AND EYE ANALYSIS** and select **Enable Jitter Measurement Based on N-Unit Interval** checkbox to enable this configuration. **TIE:** This involves Jitter analysis based on Time Interval Error (TIE). **N-Unit interval:** N-Unit Interval(N-UI) jitter analysis refers to analyzing various jitter components based on the intervals of N bits or N Unit Intervals. N-UI and Period jitter can be performed on clock or 1/0 data waveforms only. Because N-UI and period jitter are measured from edge to edge, no clock recovery is required. It is useful for capturing low-frequency jitter components and for better understanding of correlated jitter. **Period:** This is the same as N-Unit Interval based jitter measurement when N=2.
Where N =	Sets the value of N for unit inverval. Available when **Base Jitter Measurement Technique** is set to **N-Unit Interval**. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.
Pattern Detection	Set the pattern detection method as **Auto** or **Manual**. Auto method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. Manual method allows (and requires) that the Pattern Type and associated controls be set manually.
Pattern Type	Set the pattern type as **Repeating** or **Arbitrary**. If the data signal is repeating pattern of N bits, then Repeating pattern type should be selected. If the data signal is non-repeating pattern or unknown then Arbitrary pattern type should be selected. Available when **Pattern Detection** is **Auto**.
Pattern Length	Enter the pattern length. Default length is 2 UI. When Pattern Detection is Auto, this field shows the detected pattern length. When Pattern Detection is Manual, this control must be set to match the actual pattern length. If the manually-set pattern length is inconsistent with the detected pattern length, process stops but a warning will be logged. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Repeating**.
Window Length	Enter the window length. Default length is 10 UI. Determines the number of unit intervals over which pattern correlation effects are analyzed. The window should be set to a large enough value that the impulse response of the serial data transmitter and channel have settled. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Arbitrary**.

TJ@BER measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Base Jitter Measurement Technique	Sets the Base Jitter Measurement Technique as **TIE**, **Period**, or **N-Unit Interval**. Tap **Utility > User Preference > JITTER AND EYE ANALYSIS** and select **Enable Jitter Measurement Based on N-Unit Interval** checkbox to enable this configuration. **TIE:** This involves Jitter analysis based on Time Interval Error (TIE). **N-Unit interval:** N-Unit Interval(N-UI) jitter analysis refers to analyzing various jitter components based on the intervals of N bits or N Unit Intervals. N-UI and Period jitter can be performed on clock or 1/0 data waveforms only. Because N-UI and period jitter are measured from edge to edge, no clock recovery is required. It is useful for capturing low-frequency jitter components and for better understanding of correlated jitter. **Period:** This is the same as N-Unit Interval based jitter measurement when N=2.
Signal Type	Set the signal type as **Clock**, **Data**, or **Auto**. This configuration is not available when **Base Measurement Technique** is set to other than **TIE**.
Clock Edge	Set the edge of the clock signal. Available clock edges are rising, falling, and both. Available when **Signal Type** is **Clock** or **Auto**. This configuration is not available when **Base Measurement Technique** is set to other than **TIE**.
Where N =	Sets the value of N for unit inverval. Available when **Base Jitter Measurement Technique** is set to **N-Unit Interval**. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.
Pattern Detection	Set the pattern detection method as **Auto** or **Manual**. Auto method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. Manual method allows (and requires) that the Pattern Type and associated controls be set manually.

Field or control	Description
Pattern Type	Set the pattern type as **Repeating** or **Arbitrary**. If the data signal is repeating pattern of N bits, then Repeating pattern type should be selected. If the data signal is non-repeating pattern or unknown then Arbitrary pattern type should be selected. Available when **Pattern Detection** is **Auto**.
Pattern Length	Enter the pattern length. Default length is 2 UI. When Pattern Detection is Auto, this field shows the detected pattern length. When Pattern Detection is Manual, this control must be set to match the actual pattern length. If the manually-set pattern length is inconsistent with the detected pattern length, process stops but a warning will be logged. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Repeating**.
Window Length	Enter the window length. Default length is 10 UI. Determines the number of unit intervals over which pattern correlation effects are analyzed. The window should be set to a large enough value that the impulse response of the serial data transmitter and channel have settled. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Arbitrary**.
Target BER	Sets the Bit Error Rate, thereby setting the statistical level at which total jitter and eye opening are reported. Default value is 1e-12. The minimum value is 1e-18 and maximum value is 1e-2. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.

Jitter Summary measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Base Jitter Measurement Technique	Sets the Base Jitter Measurement Technique as TIE, Period, or N-Unit Interval. Tap Utility > User Preference > JITTER AND EYE ANALYSIS and select Enable Jitter Measurement Based on N-Unit Interval checkbox to enable this configuration. TIE: This involves Jitter analysis based on Time Interval Error (TIE). N-Unit interval: N-Unit Interval(N-UI) jitter analysis refers to analyzing various jitter components based on the intervals of N bits or N Unit Intervals. N-UI and Period jitter can be performed on clock or 1/0 data waveforms only. Because N-UI and period jitter are measured from edge to edge, no clock recovery is required. It is useful for capturing low-frequency jitter components and for better understanding of correlated jitter. Period: This is the same as N-Unit Interval based jitter measurement when N=2.

Field or control	Description
Signal Type	Set the signal type as Clock, Data, or Auto. This configuration is not available when Base Measurement Technique is set to other than TIE.
Clock Edge	Set the edge of the clock signal. Available clock edges are rising, falling, and both. Available when **Signal Type** is **Clock** or **Auto**. This configuration is not available when Base Measurement Technique is set to other than TIE.
Where N =	Sets the value of N for unit inverval. Available when Base Jitter Measurement Technique is set to N-Unit Interval. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.
Pattern Detection	Set the pattern detection method as Auto or Manual. Auto method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. Manual method allows (and requires) that the Pattern Type and associated controls be set manually.
Pattern Type	Set the pattern type as Repeating or Arbitrary. If the data signal is repeating pattern of N bits, then Repeating pattern type should be selected. If the data signal is non-repeating pattern or unknown then Arbitrary pattern type should be selected. Available when **Pattern Detection** is **Auto**.
Pattern Length	Enter the pattern length. Default length is 2 UI. When Pattern Detection is Auto, this field shows the detected pattern length. When Pattern Detection is Manual, this control must be set to match the actual pattern length. If the manually-set pattern length is inconsistent with the detected pattern length, process stops but a warning will be logged. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Repeating**.
Window Length	Enter the window length. Default length is 10 UI. Determines the number of unit intervals over which pattern correlation effects are analyzed. The window should be set to a large enough value that the impulse response of the serial data transmitter and channel have settled. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Arbitrary**.
Target BER	Sets the Bit Error Rate, thereby setting the statistical level at which total jitter and eye opening are reported. Default value is 1e-12. The minimum value is 1e-18 and maximum value is 1e-2. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.

Field or control	Description
Set Levels In	Sets the units used for the Eye Height@BER measurement to **%** or **Absolute**. This setting defines the horizontal measurement position using either a percentage of the unit interval or an absolute time value. Available only when Jitter Analysis Method under User Preferences is set to **Jitter + Noise**.
Measure at 'X' Seconds	Specifies the horizontal position at which the Eye Height@BER measurement is taken, in seconds. Available when Set Levels In is set to **Absolute**. Available only when Jitter Analysis Method under User Preferences is set to **Jitter + Noise**.
Measure at % of the Unit Interval	Specifies the horizontal position at which the Eye Height@BER measurement is taken as a percentage of the unit interval (UI). Available when Set Levels In is set to **%**. Available only when Jitter Analysis Method under User Preferences is set to **Jitter + Noise**.

Phase Noise measurement: Configure panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Source 2	Select the input signal channel from the Source 2 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Upper Noise Integration Limit	Sets the upper noise integration limit.
Lower Noise Integration Limit	Sets the lower noise integration limit.
Smoothing Filter	Toggles the smoothing filter On or Off to smoothen the phase noise plot. The default is **On**.

RJ δδ, PJ, RJ, DJ, J2, and J9 measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

Field or control	Description
Base Jitter Measurement Technique	Sets the Base Jitter Measurement Technique as **TIE**, **Period**, or **N-Unit Interval**. Tap **Utility > User Preference > JITTER AND EYE ANALYSIS** and select **Enable Jitter Measurement Based on N-Unit Interval** checkbox to enable this configuration. **TIE:** This involves Jitter analysis based on Time Interval Error (TIE). **N-Unit interval:** N-Unit Interval(N-UI) jitter analysis refers to analyzing various jitter components based on the intervals of N bits or N Unit Intervals. N-UI and Period jitter can be performed on clock or 1/0 data waveforms only. Because N-UI and period jitter are measured from edge to edge, no clock recovery is required. It is useful for capturing low-frequency jitter components and for better understanding of correlated jitter. **Period:** This is the same as N-Unit Interval based jitter measurement when N=2.
Signal Type	Set the signal type as **Clock**, **Data**, or **Auto**. This configuration is not available when **Base Measurement Technique** is set to other than **TIE**.
Clock Edge	Set the edge of the clock signal. Available clock edges are rising, falling, and both. Available when **Signal Type** is **Clock** or **Auto**. This configuration is not available when **Base Measurement Technique** is set to other than **TIE**.
Where N =	Sets the value of N for unit inverval. Available when **Base Jitter Measurement Technique** is set to **N-Unit Interval**. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.
Pattern Detection	Set the pattern detection method as **Auto** or **Manual**. Auto method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. Manual method allows (and requires) that the Pattern Type and associated controls be set manually.
Pattern Type	Set the pattern type as **Repeating** or **Arbitrary**. If the data signal is repeating pattern of N bits, then Repeating pattern type should be selected. If the data signal is non-repeating pattern or unknown then Arbitrary pattern type should be selected. Available when **Pattern Detection** is **Auto**.

Field or control	Description
Pattern Length	Enter the pattern length. Default length is 2 UI. When Pattern Detection is Auto, this field shows the detected pattern length. When Pattern Detection is Manual, this control must be set to match the actual pattern length. If the manually-set pattern length is inconsistent with the detected pattern length, process stops but a warning will be logged. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Repeating**.
Window Length	Enter the window length. Default length is 10 UI. Determines the number of unit intervals over which pattern correlation effects are analyzed. The window should be set to a large enough value that the impulse response of the serial data transmitter and channel have settled. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Arbitrary**.

DJ-DD measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Base Jitter Measurement Technique	Sets the Base Jitter Measurement Technique as **TIE**, **Period**, or **N-Unit Interval**. Tap Utility > User Preference > JITTER AND EYE ANALYSIS and select Enable Jitter Measurement Based on N-Unit Interval checkbox to enable this configuration. TIE: This involves Jitter analysis based on Time Interval Error (TIE). N-Unit interval: N-Unit Interval(N-UI) jitter analysis refers to analyzing various jitter components based on the intervals of N bits or N Unit Intervals. N-UI and Period jitter can be performed on clock or 1/0 data waveforms only. Because N-UI and period jitter are measured from edge to edge, no clock recovery is required. It is useful for capturing low-frequency jitter components and for better understanding of correlated jitter. Period: This is the same as N-Unit Interval based jitter measurement when N=2.
Signal Type	Set the signal type as Clock, Data, or Auto. This configuration is not available when **Base Measurement Technique** is set to other than **TIE**.
Clock Edge	Set the edge of the clock signal. Available clock edges are rising, falling, and both. Available when **Signal Type** is **Clock** or **Auto**. This configuration is not available when **Base Measurement Technique** is set to other than **TIE**.

Field or control	Description
Where N =	Sets the value of N for unit inverval. Available when **Base Jitter Measurement Technique** is set to **N-Unit Interval**. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard.
Pattern Detection	Set the pattern detection method as Auto or Manual. Auto method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. Manual method allows (and requires) that the Pattern Type and associated controls be set manually.
Pattern Type	Set the pattern type as Repeating or Arbitrary. If the data signal is repeating pattern of N bits, then Repeating pattern type should be selected. If the data signal is non-repeating pattern or unknown then Arbitrary pattern type should be selected. Available when **Pattern Detection** is **Auto**.
Pattern Length	Enter the pattern length. Default length is 2 UI. When Pattern Detection is Auto, this field shows the detected pattern length. When Pattern Detection is Manual, this control must be set to match the actual pattern length. If the manually-set pattern length is inconsistent with the detected pattern length, process stops but a warning will be logged. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Repeating**.
Window Length	Enter the window length. Default length is 10 UI. Determines the number of unit intervals over which pattern correlation effects are analyzed. The window should be set to a large enough value that the impulse response of the serial data transmitter and channel have settled. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Arbitrary**.
Target BER	Sets the Bit Error Rate, thereby setting the statistical level at which total jitter and eye opening are reported. Default value is 1e-12. The minimum value is 1e-18 and maximum value is 1e-2. Available only **Jitter Analysis Method** is Set to **Jitter Only**.

SJ@Freq measurement: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Signal Type	Set the signal type as **Clock**, **Data**, or **Auto**.

Field or control	Description
Clock Edge	Set the edge of the clock signal. Available clock edges are rising, falling, and both. Available when **Signal Type** is **Clock** or **Auto**.
Pattern Detection	Set the pattern detection method as **Auto** or **Manual**. Auto method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. Manual method allows (and requires) that the Pattern Type and associated controls be set manually
Pattern Type	Set the pattern type as **Repeating** or **Arbitrary**. If the data signal is repeating pattern of N bits, then Repeating pattern type should be selected. If the data signal is non-repeating pattern or unknown then Arbitrary pattern type should be selected. Available when **Pattern Detection** is **Auto**.
Pattern Length	Enter the pattern length. Default length is 2 UI. When Pattern Detection is Auto, this field shows the detected pattern length. When Pattern Detection is Manual, this control must be set to match the actual pattern length. If the manually-set pattern length is inconsistent with the detected pattern length, process stops but a warning will be logged. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Repeating**.
Window Length	Enter the window length. Default length is 10 UI. Determines the number of unit intervals over which pattern correlation effects are analyzed. The window should be set to a large enough value that the impulse response of the serial data transmitter and channel have settled. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Arbitrary**.
SJ Frequency	Sets the frequency for the sinusoidal jitter measurement. Default frequency is 10 MHz.
SJ Bandwidth	Sets the narrow bandwidth for sinusoidal jitter measurement. 1 MHz is considered the optimal higher limit. The default bandwidth is set to 500 kHz.

SRJ, NPJ, F/2, F/4, and F/8 measurements: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

Field or control	Description
Signal Type	Set the signal type as **Clock**, **Data**, or **Auto**.
Clock Edge	Set the edge of the clock signal. Available clock edges are rising, falling, and both. Available when **Signal Type** is **Clock** or **Auto**.
Pattern Detection	Set the pattern detection method as **Auto** or **Manual**. Auto method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. Manual method allows (and requires) that the Pattern Type and associated controls be set manually.
Pattern Type	Set the pattern type as **Repeating** or **Arbitrary**. If the data signal is repeating pattern of N bits, then Repeating pattern type should be selected. If the data signal is non-repeating pattern or unknown then Arbitrary pattern type should be selected. Available when **Pattern Detection** is **Auto**.
Pattern Length	Enter the pattern length. Default length is 2 UI. When Pattern Detection is Auto, this field shows the detected pattern length. When Pattern Detection is Manual, this control must be set to match the actual pattern length. If the manually-set pattern length is inconsistent with the detected pattern length, process stops but a warning will be logged. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Repeating**.
Window Length	Enter the window length. Default length is 10 UI. Determines the number of unit intervals over which pattern correlation effects are analyzed. The window should be set to a large enough value that the impulse response of the serial data transmitter and channel have settled. Double-tap in the field to change the value using a connected keyboard or the virtual keyboard. Available when **Pattern Type** is **Arbitrary**.

TIE measurement: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Signal Type	Set the signal type as **Clock**, **Data**, or **Auto**.

Field or control	Description
Clock Edge	Set the edge of the clock signal. Available clock edges are rising, falling, and both. Available when **Signal Type** is **Clock** or **Auto**.

Noise Measurement Panel

NOTE: Noise Measurements are only available with DJAN option.

TN@BER, Noise Summary measurement: Configuration panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. Use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Signal Type	Sets the signal type as **Clock**, **Data**, or **Auto**.
Pattern Detection	Sets the pattern detection method as **Auto** or **Manual**. **Auto** method causes the data pattern to be detected automatically on the first acquisition following a configuration change. Based on this detection, the Pattern Type and associated controls are then configured optimally for the given record length. **Manual** method allows (and requires) that the Pattern Type and associated controls be set manually.
Pattern Type	Sets the pattern type to **Repeating** or **Arbitrary**. Select **Repeating** when the data signal consists of a repeating pattern of N bits. Select **Arbitrary** when the data signal is non-repeating or the pattern is unknown. Available when Pattern Detection is set to **Auto**.
Pattern Length	Specifies the pattern length. The default value is **2 UI**. When Pattern Detection is set to **Auto**, this field displays the detected pattern length. When Pattern Detection is set to **Manual**, set this value to match the actual pattern length. If the manually entered pattern length does not match the detected pattern length, processing stops and a warning is logged. Double-tap the field to enter a value using a connected keyboard or the virtual keyboard. Available when Pattern Type is set to **Repeating**.

Field or control	Description
Window Length	Specifies the window length. The default value is **10 UI**. Determines the number of unit intervals over which pattern correlation effects are analyzed. Set the window length large enough to allow the impulse response of the serial data transmitter and channel to settle. Double-tap the field to enter a value using a connected keyboard or the virtual keyboard. Available when Pattern Type is set to **Arbitrary**.
Target BER	Sets the Bit Error Rate (BER), which determines the statistical level at which total jitter and eye opening are reported. The default value is **1e-12**. The minimum value is **1e-18** and maximum value is **1e-2**.
Set Levels In	Sets the levels as % or Absolute. It defines the exact time in percentage terms or absolute values.
Measure at 'X' Seconds	Sets the horizontal position where the measurement is taken in terms of seconds. Available when Set Levels In is **Absolute**
Measure at % of the Unit Interval	Sets the horizontal position at which the measurement is taken as a percentage of the unit interval. Available when Set Levels In is set to **%**.

Time measurement panel

SSC Freq Dev measurement: Configure panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Bitrate Detection	Set the bitrate to **Auto** or **Manual**. Default is Auto. Auto bitrate detection enables the automatic detection of the data rate on the first acquisition following a configuration change. In Manual configuration, allows you to specify the approximate data rate in bits per second (b/s).
Detected Bitrate	Auto bitrate is 2.5 Gb/s. The readout displayed when **Bitrate Detection** is **Auto**. Enter the bitrate value manually using keyboard or double-tap and enter the value using virtual keypad. Available when **Bitrate Detection** is **Manual**.

SSC Modultion Rate measurement: Configure panel

Field or control	Description
Source 1	Select the input signal channel from the Source 1 fields.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.

Reference Levels panel (Jitter 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. Hysteresis settings are critical for PWM edge extraction.

See Reference Levels panel (Measurement configuration menu) (on page 303) for the Reference panel fields and controls.

Other measurement settings panels

Jitter measurement configuration menu overview (on page 262)

Jitter Measurement Name panel (Measurement configuration menu) (on page 263)

Configure panel (Jitter measurement configuration menu) (on page 264)

Clock Recovery panel (Jitter measurement configuration menu) (on page 285)

Filter/Limit Results panel (Jitter measurement configuration menu) (on page 286)

Gating panel (Jitter measurement configuration menu) (on page 286)

Pass/Fail Testing panel (Jitter measurement configuration menu) (on page 286)

Clock Recovery panel (Jitter 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.

NOTE: Clock recovery is not applicable when Base Measurement Technique is set to anything other than TIE.

See Clock Recovery panel (Measurement configuration menu) (on page 304) for the Clock Recovery panel fields and controls.

Other measurement settings panels

Jitter measurement configuration menu overview (on page 262)

Jitter Measurement Name panel (Measurement configuration menu) (on page 263)

Configure panel (Jitter measurement configuration menu) (on page 264)

Reference Levels panel (Jitter measurement configuration menu) (on page 285)

Gating panel (Jitter measurement configuration menu) (on page 286)

Filter/Limit Results panel (Jitter measurement configuration menu) (on page 286)

Pass/Fail Testing panel (Jitter measurement configuration menu) (on page 286)

Gating panel (Jitter measurement configuration menu)

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

See Gating panel (Measurement configuration menu) (on page 309) for the Gating panel fields and controls.

Other measurement settings panels

Jitter measurement configuration menu overview (on page 262)

Jitter Measurement Name panel (Measurement configuration menu) (on page 263)

Configure panel (Jitter measurement configuration menu) (on page 264)

Reference Levels panel (Jitter measurement configuration menu) (on page 285)

Clock Recovery panel (Jitter measurement configuration menu) (on page 285)

Filter/Limit Results panel (Jitter measurement configuration menu) (on page 286)

Pass/Fail Testing panel (Jitter measurement configuration menu) (on page 286)

Filter/Limit Results panel (Jitter 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 310) for the Filter/Limit Results panel fields and controls.

Other measurement settings panels

Jitter measurement configuration menu overview (on page 262)

Jitter Measurement Name panel (Measurement configuration menu) (on page 263)

Configure panel (Jitter measurement configuration menu) (on page 264)

Reference Levels panel (Jitter measurement configuration menu) (on page 285)

Clock Recovery panel (Jitter measurement configuration menu) (on page 285)

Gating panel (Jitter measurement configuration menu) (on page 286)

Pass/Fail Testing panel (Jitter measurement configuration menu) (on page 286)

Pass/Fail Testing panel (Jitter measurement configuration menu)

Use these settings to test measurement values against a specified limit and set actions for the instrument to take on failure.

See Pass/Fail Testing panel (Measurement Settings menu) (on page 311) for the Pass/Fail Testing panel fields and controls.

Other measurement settings panels

Jitter measurement configuration menu overview (on page 262)

Jitter Measurement Name panel (Measurement configuration menu) (on page 263)

Configure panel (Jitter measurement configuration menu) (on page 264)

Clock Recovery panel (Jitter measurement configuration menu) (on page 285)

Filter/Limit Results panel (Jitter measurement configuration menu) (on page 286)

Gating panel (Jitter measurement configuration menu) (on page 286)

Pass/Fail Testing panel (Jitter measurement configuration menu) (on page 286)

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.

Prerequisite: An Eye Diagram or BER Eye Contour plot with a mask enabled. See Add mask testing to an Eye Diagram plot (on page 88).

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
Total	Sets the total number of hits needed to fail an eye mask test.

Right click menu functions associated with mask segments

The following mask definition functions are available when you right click on a mask segment.

Mask Segments controls

Field or control	Description
Triangle	Sets the shape of the mask to a triangle.
Rectangle	Sets the shape of the mask to a rectangle
Trapezoid	Sets the shape of the mask to a trapezoid.
Hexagon	Sets the shape of the mask to a hexagon.
Custom	Sets the shape of your choice.
Create Duplicate	Creates a new segment with the same characteristics.
Configure Segment	Opens the Mask Definition configuration menu for the selected segment.
Delete Segment	Deletes the selected segment.
Delete All	Deletes all segments associated with the mask.

Tolerance Mask controls

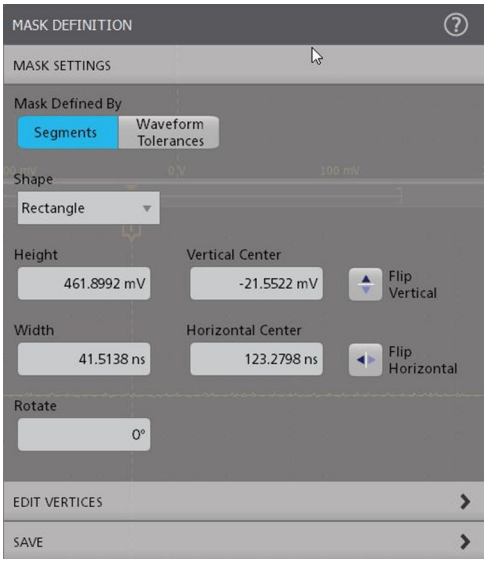
Field or control	Description
Configure Mask	Opens the Mask Definition configuration menu for the selected mask.
Delete Mask	Deletes the selected mask.

Mask Definition configuration menu

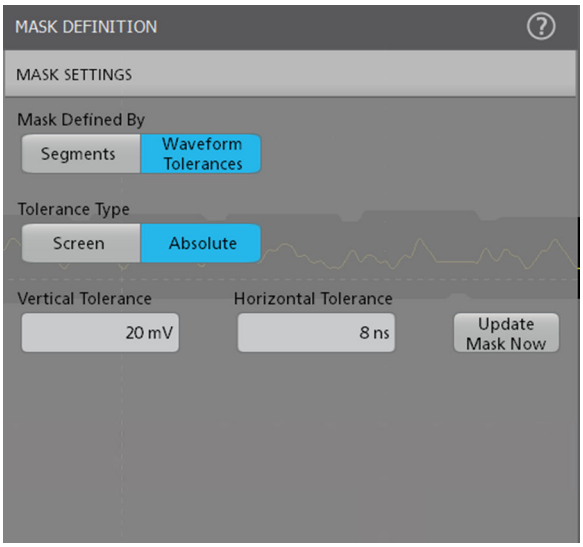
Use the Mask Definition menu to edit mask segment parameters. Double tapping a mask segment opens the Mask Definition configuration menu.

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

To create a mask segment, see Create a Mask (on page 114).



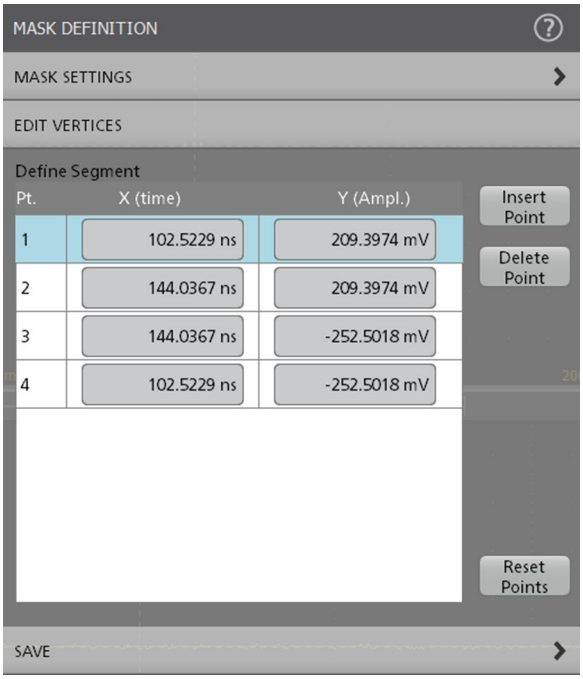
Waveform Mask Settings panel fields and controls



Waveform Mask Settings panel fields and controls

Field or control	Description
Mask Defined By	Define the way you edit the mask. The default option is **Segments**.
Shape	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 segment, and then does a best-fit approach to create the specified shape. If you made changes to a default shaped segment that results in an segment that no longer meets the default shape definition, that segment is listed as a Custom shape in the menu.
Height	Sets the segment height, in amplitude units, between the topmost vertex and the bottommost vertex.
Vertical Center	Sets the segment vertical center, in amplitude units, as the point halfway between the topmost vertex and bottommost vertex.

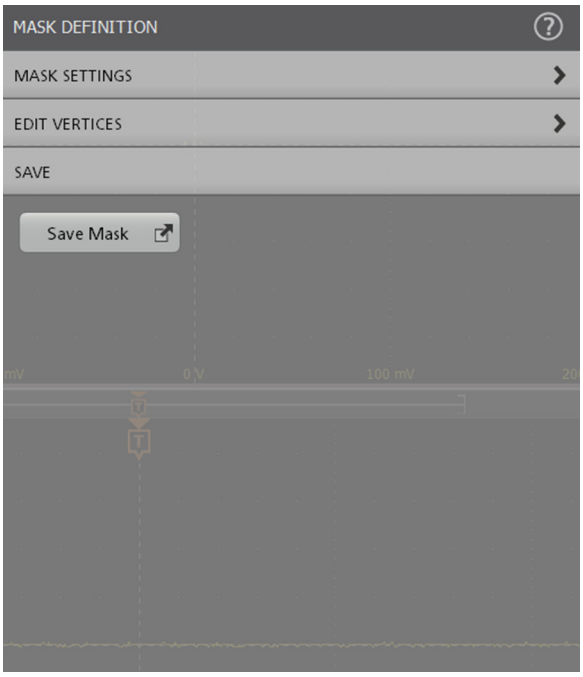
Field or control	Description
Flip Vertical	Flips the segment vertically around its Vertical Center value.
Width	Sets the segment width, in time units, between the leftmost vertex and the rightmost vertex.
Horizontal Center	Sets the segment horizontal center, as time units, as the point halfway between the leftmost vertex and rightmost vertex.
Flip Horizontal	Flips the segment horizontally around its Horizontal Center value.
Rotate	Rotates the segment in units of degrees, from 0° to 360°. The rotation is an absolute angle measurement referenced from 0°, where 0° is the position of the segment when it was first created. For example, if you rotate the segment 40°, and then rotate it again with 20°, the resulting segment rotation is 20°. The segment height and width are relative to the current segment orientation, and automatically change places as needed when the segment is rotated.
Tolerance Type	Sets the tolerance type as **Screen** or **Absolute**. Default type is screen.
Vertical Tolerance	Tap the Vertical Tolerance field and enter the tolerance value as divisions of the current channel settings (volts, amps, and so on). Or double-tap on the field and use the A knob to change the value.
Horizontal Tolerance	Tap the Horizontal Tolerance field and enter the tolerance value as divisions of the current channel settings (volts, amps, and so on). Or double-tap on the field and use the A knob to change the value.
Update Mask Now	Updates the mask as per the configured tolerance values.



Edit Vertices panel fields and controls

Field or control	Description
Define Segment	A table that lists the X (time) and Y (amplitude) values for each vertex (point) of the segment. Use the scroll bar to show points if there are more than nine points in the segment. Selecting a row in the table highlights the associated vertex on the segment. Use the Multipurpose knobs to change the value of the X or Y settings, or double-tap a setting and enter the value directly.
Insert Point	Inserts a new row above the selected row and creates a new vertex on the segment shape. The new vertex is halfway between the vertices defined in the prior row and the following row in the table.
Delete Point	Deletes the currently selected point, keeps the row selected, and moves all rows below it up one row. The **Delete Point** button is not available when a triangular segment is selected.
Reset Points	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.

Redo and Undo are available for most Edit Vertices panel controls.



Save panel fields and controls

Field or control	Description
Save Mask	Saves the mask in the desired location with the given file name.

Mask Test badge configuration menu

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

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

Mask Test badge menu

Field or control	Description
Mask Test	Sets the mask test as **Off**, **Mask Hits**, or **Mask Margin**.
Pass/Fail Criteria	Defines the pass/fail criteria for the mask test. Define the **Total Hits** and **Hit Ratio** to set this criteria. Available when **Mask Test** is **Mask Hits**. Mask Hit Ratio is the ratio between total UI hits across all mask segments and total number of UI rendered on an eye diagram. The Mask Hit Ratio is expressed in %. When **Auto Fit Mask to Minimize Hits** box is checked, the Mask Hit Ratio is measured as the ratio between the total number of sample hits and the total number of rendered samples. The plot mask badge displays the result as Hit Ratio. Mask Hit Ratio is only applicable when the **Eye Rendering** is **Complete Vectors** or **Complete Dots**. If **Hit Ratio** is selected and **Eye Rending** is set to **Fast**, it displays below information. Mask Hit Ratio is not compatible with Fast Eye Rendering. Enabling Mask Hit Ratio will switch to Complete Eye Rendering. Would you like to proceed? Tap **Yes, enable Mask Hit Ratio** to switch the **Eye Rendering** mode to **Complete Vectors**. If you select **No, maintain Fast Rendering**, the pass/fail criteria will remain at Total hits and the Hit Ratio results are not displayed in the mask badge. If the **Mask Hit Ratio** checkbox is selected and you attempt to switch the **Complete Vectors** or **Complete Dots** setting to **Fast**, it displays below warning. **WARNING:** Fast Eye Rendering is not compatible. with Mask Hit Ratio. Turning Mask Hit Ratio off. When the warning occurs the pass/fail criteria changes to Total Hits and the Eye Rendering mode changes to Fast. No Hit Ratio results are displayed in the mask badge.
Fail If Greater Than	Sets the threshold to use when determining if the mask test is pass or fail. The units depend on pass/fail criteria. Selecting **Total Hits** allows you to enter an integer value and **Hit Ratio** allows you to enter the value as a percentage. Available when **Mask Test** is **Mask Hits**.
Auto Fit Mask to Minimize Hits	Enables the auto fit algorithm to adjust the mask horizontally to minimize the mask hits. A positive offset indicates a shift to the right. A negative offset indicates a shift to the left. The offset value is displayed in the eye diagram plot. Available when **Mask Test** is **Mask Hits** or **Mask Margin**.
Mask Movement Range (UI) **From: To:**	Sets the horizontal mask movement range when **Auto Fit Mask to Minimize Hits** is enabled. Available when **Mask Test** is **Mask Hits** or **Mask Margin** and **Auto Fit Mask to Minimize Hits** box is checked.

Field or control	Description
Mask Expansion Stop Criteria	Specifies when the mask expansion process should stop through the **Total Hits** and **Hit Ratio** settings. Available when **Mask Test** is **Mask Margin**. **Total Hits** allows you to specify a fixed number of allowable mask hits. The mask expansion proceeds until this count is reached. This is useful when you want to limit the absolute number of mask violations when expanding. **Hit Ratio** allows the mask expansion to proceed until the ratio of mask violations to total number of samples in the eye diagram reaches the specified threshold. The input is provided as a percentage.
Stop When Greater Than	Enter the threshold value for the stop criteria. For **Total Hits**, enter a positive integer. For **Hit Ratio**, enter a decimal percentage value. Available when **Mask Test** is **Mask Margin**.
Mask Expansion Resolution	Sets the resolution that determines the accuracy of the mask margin. Available when **Mask Test** is **Mask Margin**.
Fail if Less Than	Specifies the minimum acceptable margin value as a percentage for the test to pass. If the margin is less than the specified threshold, the test result displays as Fail. Available when **Mask Test** is **Mask Margin**.

Guidelines for mask margin

- The Mask Margin indicates the amount a mask can expand before it intersects with the eye diagram.
- The algorithm considers the center-mask polygon as the reference for the expansion. This polygon is uniformly scaled in both vertical and horizontal directions, preserving the slope of the mask edges throughout the expansion.
- The algorithm iteratively adjusts the expansion until the resulting margin meets the defined precision and the Mask Expansion Stop Criteria settings. For example, a 20% mask margin means that no violations occurred after the original mask is scaled by 20%. If the input mask already meets the stop criteria, the algorithm contracts the mask instead, resulting in a negative margin value. Expanded masks are displayed in green, while contracted masks appear in red. The original mask get displays in grey.
- The stop criteria for mask expansion is configured in two ways: based on the total number of hits across all segments or by the hit ratio. A mask margin test status is indicated as either pass or fail based on the specified threshold value in the **Fail if Less Than** field. If the reported margin falls below the threshold, the test is displayed as failed.

Right click menu functions associated with the Mask Test badge

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

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

Math configuration menu overview

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

To access a Math configuration menu, double-tap a **Math** waveform badge on the Settings bar. If no Math badge is present, tap the **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 293)

Equation Editor (Math configuration menu) (on page 297)

Math configuration menu

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

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

Math configuration menu fields and controls

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

Field or control	Description
Math Type	Sets the type of math waveform to display. **Basic** creates a math waveform by adding, subtracting, multiplying, or dividing two analog waveforms. **FFT** opens an FFT view of the specified signal to display the frequency components of that signal. Double-tap on the FFT view to open its configuration menu. **Advanced** displays a drop-down from which to select the 20 last-accessed equations created by the Equation Editor. This mode also provides access to the Equation Editor. See Equation Editor (Math configuration menu) (on page 297).
Source, Source1, Source 2	Defines the signal source or sources for a **Basic**, **FFT** math waveform. Available when **Math Type** is **Basic**, **FFT**.
Basic math operation list	Located between the **Source 1** and **Source 2** fields. A drop-down list to select a basic math operation (add, subtract, multiply, divide) to apply to the two sources. Available when **Math Type** is **Basic**.
Interpolation	Select to display the record points between the sampled points. Default is unchecked. Available when **Math Type** is **Basic** and **Source 1** is **Bus**. Interpolation uses Sin(x)/x method. Sin(x)/x connects using a Sin(x)/x curve between the computed values. This makes the gain or phase curve smoother than linear interpolation.
Field	Select the field for the selected bus. Default is Data. Available when **Math Type** is **Basic** and **Source 1** is **Bus**.
Signed Data	Sets the data as signed decimal. Default is unchecked.
Math n =	Lists the last-accessed advanced equation. Tap the down arrow to display a list of the last-accessed equations (up to a maximum of 20) created by the **Equation Editor**. Select an equation to display that math waveform. Tap **Edit** to open the **Equation Editor** to edit the displayed equation. You can also double-tap on an equation in this field and directly edit the equation using the virtual keyboard. See Equation Editor (Math configuration menu) (on page 297). Available when **Math Type** is **Advanced**.
Edit	Opens the **Equation Editor** to create advanced math waveforms from analog channels, reference, math waveform, measurement, and variable sources. Tap the **Edit** button to open the **Equation Editor**. See Equation Editor (Math configuration menu) (on page 297). Available when **Math Type** is **Advanced**.

Math waveform guidelines

- You can take measurements on math waveforms in the same way as on channel waveforms.
- Math waveforms derive their horizontal scale and position from the sources in their math expressions. Adjusting these controls for the source waveforms also adjusts the math waveform.
- You can Zoom on math waveforms.

Create/Edit filter configuration menu

Use the create/edit filter menu to design IIR or FIR filter for a given signal. This menu also allows you to save the configured filter into a file and recall in the future.

To access the Create/Edit 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 **Filter**.
3. Tap **Create/Edit Filter** to open the Create/Edit menu.

If the filter is created successfully for the first time or any filter file is loaded previously, the control **Create Filter** is changed to **Edit Filter**.

Field or Control	Description
Filter Type	Select the required filter type from the drop-down. **Low-Pass**, **High-Pass**, **Band-Pass**, **Band-Stop**, **All-Pass**, **Hilbert**, **Differentiator**, **Raised Cosine**, **Root-Raised Cosine**.
Filter Response	Select the required filter response from the drop-down. **Butterworth**, **Chebyshev I**, **Chebyshev II**, **Elliptical**, **Gaussian**, **Bessel-Thompson**, **Custom**.
Filter Order	Enter the filter order value in the text field. Supported filter order value for IIR is 8 and below and for FIR is 5001 and below. Filter order is auto-computed for custom filter.
Cutoff frequency	Enter the cutoff frequency value in the text field. Available only when **Filter Type** is **Low Pass** or **High-Pass**
Low Cutoff Frequency	Enter the low cutoff frequency value in the text field. Available only when **Filter Type** is **Band-Pass** or **Band-Stop**
High Cutoff Frequency	Enter the high cutoff frequency value in the text field. Available only when **Filter Type** is **Band-Pass** or **Band-Stop**
Pass Band Ripple	Enter the Pass Band Ripple value in the text field. Available only when **Filter Response** is **Chebyshev I** or **Elliptical**
Transition Width	Enter the transition width value in the text field. Default value is 15 MHz. For low-pass filters the transition width cannot exceed (0.45*SR) - cutoff -1. For high-pass filters it cannot exceed the cutoff frequency. For band-pass and band-stop, the transition width is minimum of lower cutoff or (0.5 * SR - high cutoff). Available only when **Filter Response** is **Custom**.
Stop Band Attenuation	Enter the stop band attenuation value in the text field. Available only when **Filter Response** is **Chebyshev II** or **Elliptical**

Field or Control	Description
Delay	Enter the delay value. The unit is in seconds. Delay in seconds introduces the appropriate time delay on the filtered output MATH waveform which is equivalent to the phase shift in the frequency domain.
Standard Deviation	The standard deviation defines the Gaussian filter window width.
Symbols	Enter the number of symbols in the text field by which the filter span. Symbols value is >0. Default value is 1. Available only when **Filter Type** is **Raised Cosine** or **Root-Raised Cosine**
Symbol Duration	Enter the symbol duration value in the text field. Symbol duration is >0. Default value is 10 ns. **NOTE:** Enter input data signal period for proper filter response. Available only when **Filter Type** is **Raised Cosine** or **Root-Raised Cosine**
Roll-off Factor	Enter the roll-off factor value in the text field. Roll off factor is a floating number between 0 to 1. Default value is 1. Available only when **Filter Type** is **Raised Cosine** or **Root-Raised Cosine**
Filter response image	Displays the selected filter responses. ▪ Magnitude and Phase Response ▪ Impulse Response ▪ Step Response
Apply Filter	Tap to apply filter, generated on the input source and filtered waveform is shown in MATH. This button is disabled after you apply the filter by tapping **Apply Filter**, until a new filter is generated. **NOTE:** If MATH waveform is empty then filter is not applied and error is shown. You may need to adjust the S/R or any filter design parameters accordingly.
Load Filter File	Tap to recall the previously saved filter file. In Load Filter window, you can select **View filter response image** to recall filter response along with the filter file. Supported filter file format is .flt.
Save Filter To File	Tap to save the created filter into a file for future use. In Save Filter window, user can select **Save filter response image** to save filter response along with the filter file. Supported filter file format is .flt.
Generate Filter	Tap to generate the filter and to update the filter response. This button is disabled after you generate the filter by tapping **Generate Filter**, until any changes are made to the configurations.
Abort	Tap to stop or cancel the filter design during filter designing flow.

See also

User Defined Filter (on page 574)

User defined filter errors and warnings (on page 615)

Equation Editor (Math configuration menu)

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

To access the math Equation Editor:

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

Equation Editor menu fields and controls

Field or control	Description
Sources	Lists all available sources that you can add to an equation. Tap a source icon to add it to the cursor position in the Math x = input box. 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.
Functions	Select the math functions to apply to your signal or signals. See Add Functions (math Equation Editor) (on page 299).
Keypad	Use to enter numeric, basic math operations, and logic conditions.
Math <x> =	The field that lists the equation components that you add with the equation controls for the numbered Math waveform. You can also directly edit this field.
Clear	Clears the math equation field.
Arrows	Use to move the cursor position in the **Math <x> =** equation field.
Delete	Deletes the selected part of the equation (recommend that you use a mouse to more easily select equation text).
Backspace	Deletes the character to the left of the cursor.
Apply	Tap to apply the math expression shown in the **Math <x> =** field and display the math waveform. The first ~ 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.
Cancel	Closes the editor menu, does not update the equation list if you made any changes from the last time you Applied or OK'd an equation.
OK	Applies the equation to the math waveform, closed the Advanced Editor window, and adds the equation to the available equation list.

Equation editor guidelines

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

Add Filter menu (math Equation Editor)

Use the Add Filter menu to add a high pass, low pass, or arbitrary filter expression to the Filters column of the Equation Editor Sources table.

To access the Add Filter menu:

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

Add Filter menu fields and controls

Field or control	Description
Filter Type	Sets the filter type: **High Pass**, **Low Pass**, or **ArbFlt** (arbitrary filter). ArbFlt requires you to load a FLR-format filter file. If loading a filter file, tap Load to navigate to and select the FLR file to load.
Cutoff Frequency	Sets the filter cutoff frequency as a predefined fraction of the sample rate (SR). Default is 0.25 * SR.
Label	Shows the filter selections as A:B, where A is the filter type (HP or LP) and B is cutoff frequency setting. Arbitrary filter files use the file name as the label.

Add Variable menu (math Equation Editor)

Use the Add Variable menu to add a defined variable source to the Equation Editor Sources table, which you can then add to your math waveform expression.

To open the Add Variable menu:

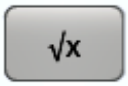
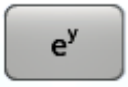
1. Double-tap a Math waveform badge on the Settings bar. If no Math badge is present, tap the **Add New Math** button to add a Math waveform and open the configuration menu.
2. Set Math Type to **Advanced**.
3. Tap **Edit** to open the Equation Editor menu.
4. Tap **Add Variable** to open the Add Variable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

Add Variable menu fields and controls

Field or control	Description
Variable Type	Selects a predefined variable to add to the Variables column of the Equation Editor. **Sample Rate** creates a variable with the value of the current sample rate. **Sample Interval** creates a variable with the value of the current sample interval (1/sample rate) **Record Length** creates a variable with the value of the current record length.
Cancel	Closes the menu without adding a variable to the **Equation Editor** Variables column.
OK	Closes the menu and adds the current variable selection to the Variables column of the **Equation Editor's Sources** table.

Add Functions (math Equation Editor)

Use the Add Functions controls to add predefined math operations to 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.
	Base 10 logarithm. Inserts the text LOG(into the math expression. Enter an argument to the function. The log function produces the base 10 logarithm of the argument.
	Square root. Inserts the text SQRT(into the math expression. Enter an argument to the function.
	Absolute. Inserts the text FABS(into the math expression. The FABS function takes the absolute value of the expression.
	Derivative. Inserts the text DIFF(into the math expression. Enter an argument to the function. The derivative function produces the derivative of the argument.
	Natural logarithm. Inserts the text LN(into the math expression. The natural logarithm function produces the natural logarithm of the argument.
	Natural antilog. Inserts the text EXP(into the math expression. The EXP function produces the natural antilog of the argument.

Button	Description
	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.

Measurement configuration menu overview

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

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

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

Measurement configuration menu fields, controls, and panels

Field, control, or panel	Description
Measurement Statistics (Measurement name panel)	A list of measurement statistics related to the measurement. You can add these statistics to a measurement badge by selecting the **Show Statistics in Badge** control.
Show Statistics in Badge (Measurement name panel)	Adds the listed statistical measurement readouts to the measurement badge readout.
Configure panel	Sets the source, label text, and other fields that are specific to each measurement type. See Configure panel (Measurement configuration menu) (on page 301).
Reference Levels panel	Sets the reference levels and units used to take measurements, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values. See Reference Levels panel (Measurement configuration menu) (on page 303).
Gating panel	Sets the measurement region (gate) used to take measurements. Select the scope of the gate setting (global or local), and the type of gating to use. See Gating panel (Measurement configuration menu) (on page 309).
Pass/Fail Testing panel	Sets the conditions for pass/fail testing for measurements and sets actions to take when a measurement fails. See Pass/Fail Testing panel (Measurement Settings menu) (on page 311).

Measurement Name panel (Measurement configuration menu)

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

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

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

Field or control	Description
Measurement Statistics	A list of measurement statistics. You can add these statistics to a measurement badge by selecting the **Show Statistics in Badge** control.
Show Statistics in Badge	Select to add the listed statistical measurement readouts to the measurement badge.

See also

Configure panel (Measurement configuration menu) (on page 301)

Reference Levels panel (Measurement configuration menu) (on page 303)

Clock Recovery panel (Measurement configuration menu) (on page 304)

Gating panel (Measurement configuration menu) (on page 309)

Filter/Limit Results panel (Measurement Settings menu) (on page 310)

Pass/Fail Testing panel (Measurement Settings menu) (on page 311)

Configure panel (Measurement configuration menu)

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

To open the Configure panel:

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

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

Field or control	Description
Source	Sets the signal source used to take the measurement. Tap the field to show the list of available sources. If the measurement requires more than one source, multiple Source fields are displayed.
Label	Sets the name of the measurement. You can use the default name, or double-tap in the field and change the label using a connected keyboard or the virtual keyboard.
Signal Type	Sets the signal type (Clock, Data, Auto, Single Ended, Differential) of the source signal for some measurements.
Edge, Clock edge	Sets the edge of the signal to use for starting the measurement.
From Edge	Sets the Source 1 waveform edge on which to start the measurement, for two-source measurements.
To Edge	Sets the Source 2 waveform edge on which to stop the measurement, for two-source measurements.
Calculate One Measurement Per	Sets the amount of waveform data to use to calculate one measurement; one measurement across the entire waveform record or one measurement for each cycle of the waveform in the record.

Other measurement panels

Measurement Name panel (Measurement configuration menu) (on page 300)

Reference Levels panel (Measurement configuration menu) (on page 303)

Clock Recovery panel (Measurement configuration menu) (on page 304)

Clock Recovery- Advanced Settings configuration menu (on page 308)

Gating panel (Measurement configuration menu) (on page 309)

Filter/Limit Results panel (Measurement Settings menu) (on page 310)

Filter/Limit Results panel (Jitter measurement configuration menu) (on page 286)

Pass/Fail Testing panel (Measurement Settings menu) (on page 311)

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

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

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

Field or control	Description
Base Top Method	Sets the method to calculate the waveform Base and Top values, which is then used to calculate the High, Mid, and Low reference levels. **Auto** is the default method, which automatically determines the most suitable Base Top method to use. It initially applies the Histogram Mode method. If the Mode value for either the top or bottom half of the waveform is unstable or ambiguous, the MinMax method is used instead. **MinMax** Uses the minimum and maximum values in the waveform record to determine the base and top amplitude. Useful on a waveform with low noise and free from excessive overshoot. **Histogram Mean** uses histogram analysis to calculate the mean or average value using all values above and below the waveform midpoint. Top is set to the mean high value, and Base is set to the mean low value. **Histogram Mode** uses histogram analysis to calculate the most common values above and below the waveform midpoint. Top is set to the common high value, and Base is set to the common low value. **Histogram Eye Center** uses histogram analysis to determine the base top amplitude. Creates a histogram of the amplitudes in the center of each bit (unit interval) while ignoring the waveform during bit transitions. The histogram should have a peak at the nominal high level and another peak at the nominal low level.
Hysteresis	Sets the threshold margin to the reference level which the signal must cross to be recognized as changing; the margin is the relative reference level plus or minus half the hysteresis. Use hysteresis to filter out spurious events. Tap the field and use the Multipurpose Knob to change the value.

Other measurement settings panels

Measurement Name panel (Measurement configuration menu) (on page 300)

Configure panel (Measurement configuration menu) (on page 301)

Gating panel (Measurement configuration menu) (on page 309)

Pass/Fail Testing panel (Measurement Settings menu) (on page 311)

Clock Recovery panel (Measurement configuration menu)

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

To access the Clock Recovery panel:

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

Measurement configuration menu, Clock Recovery panel

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

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

Field or control	Description
PLL Model	Select the PLL model type. The PLL control area provides control over the phase-locked loop used for clock recovery. You can choose the loop bandwidth and the loop order, and if a Type II loop is chosen, you can specify the damping factor. To set the loop bandwidth automatically, based on a serial standard, select PLL: Standard BW as the clock recovery method. From the Standard: b/s list box, select the standard that matches your data link. For example, choose “PCI-E: 2.5” to test a 2.5 Gbit/second PCI Express link. In this case, the PLL bandwidth will be set to 1.5 MHz, which is 1/1667 of the baud rate as specified in PCI Express standard. You can use the PLL Model list box to choose between Type I and Type II loop. A Type I loop has a transfer function that approaches zero frequency with a slope of 1/s and a Type II loop approaches zero frequency with a 1/s2 slope (In much of the PLL literature, these terms are used interchangeably with First-Order and Second-Order loops. For a thorough discussion of loop type versus order, see Frequency Synthesis by Phase Lock, by William Egan). Available when **Method** is **Explicit Clock** and **Mode** is **Explicit Clock** - **PLL**.
JTF BW	Displays or sets the Jitter Transfer Function bandwidth that has been configured based on the current standard. Available when Method is **Explicit Clock** and **Mode** is **Explicit Clock** - **PLL** and **PLL Model** is **Type II**.
Damping	Sets the damping factor for the PLL. It is enabled only for Type II phase-locked loop. Available when **Method** is **Explicit Clock** and **Mode** is **Explicit Clock** - PLL and **PLL Model** is **Type II**.
Automatic Transition Density Compensation	Toggles the automatic transition density compensation to **On** or **Off**. By default it is Off. When on, the loop feedback time constants are adjusted to make the actual transfer function more closely match a mathematical filter polynomial. When off, the clock recovery algorithm is configured for an assumed 50% transition density. Available when **Method** is **PLL** or **Explicit clock** and **Mode** is **Explicit Clock-PLL**.
Advanced	Opens the Clock Recovery-Advanced Settings configuration menu to refine the recovered clock signal. See Clock Recovery- Advanced Settings configuration menu (on page 308). Available when **Method** is **PLL**.

About constant clock recovery

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

About PLL loop BW versus JTF BW

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

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

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

PLL-based clock recovery

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

Other measurement settings panels

Measurement Name panel (Measurement configuration menu) (on page 300)

Configure panel (Measurement configuration menu) (on page 301)

Reference Levels panel (Measurement configuration menu) (on page 303)

Clock Recovery- Advanced Settings configuration menu (on page 308)

Gating panel (Measurement configuration menu) (on page 309)

Filter/Limit Results panel (Measurement Settings menu)

Pass/Fail Testing panel (Measurement Settings menu) (on page 311)

Clock Recovery- Advanced Settings configuration menu

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

To open the Clock Recovery-Advanced Settings configuration menu:

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

Clock Recovery-Advanced Settings configuration menu, fields and controls

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

Field or control	Description
Browse	Opens a standard file navigation window. Use to navigate to and select a pattern file.

See also

Measurement Name panel (Measurement configuration menu) (on page 300)

Configure panel (Measurement configuration menu) (on page 301)

Reference Levels panel (Measurement configuration menu) (on page 303)

Clock Recovery panel (Measurement configuration menu) (on page 304)

Gating panel (Measurement configuration menu) (on page 309)

Filter/Limit Results panel (Measurement Settings menu)

Pass/Fail Testing panel (Measurement Settings menu) (on page 311)

Gating panel (Measurement configuration menu)

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

To open the **Gating** panel:

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

Gating panel, fields and controls

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

Field or control	Description
Gating	Sets whether this measurement's gating settings are **Global** or **Local**. When **Global** is selected, changing anything in this panel causes the same change to all other measurements that also have **Global** selected. When **Local** is selected, settings in this panel only effect this measurement.
Gating Type	Sets the gate type used to take measurements. **None**: Measurements are taken across the entire record. **Screen**: Measurements are taken on the portion of the waveform shown on the display. When zoom is on, the 'display' on which to measure is the zoom window. **Cursors**: Measurements are taken on the portion of the waveform between the cursors. **Logic**: Measurements are taken only when the logical state of a specified waveform is true. **Search**: Measurements are taken between the start and stop times defined by a specified search. **Time**: Measurements are taken between the times specified in the **Start Gate Time** and **End Gate Time** fields, relative to the trigger point (0 s).

Field or control	Description
Source	Sets the signal source to use for Logic or Search gates. If **Gating Type** is **Logic**, the **Source** field lists all available sources. If **Gating Type** is **Search**, the **Source** field lists all available searches.
Threshold	Sets the threshold value for the Logic gate source to be considered a logic 1 value. Available when **Source** is **Logic**.
Hysteresis	Sets the Hysteresis value for the Logic gate source. Available when **Source** is **Logic**.
Active	Sets the logic state value for the Logic gate source. Available when **Source** is **Logic**.
Start Gate Time **End Gate Time**	Sets the starting time (Start Gate) and ending time (End Gate) between which to take a measurement. The starting and ending gate times are relative to the trigger time (0 s). Available when **Gating Type** is **Time**.

Guidelines

- Only one gated region is allowed per measurement. You can create duplicate measurements and define different gates for each one.
- Tap on a gating-enabled measurement badge to display the vertical gate bars on the waveform for that measurement.
- Gating is indicated by two vertical bars; one at the start of the gated region and one at the end.
- If the measurement has more than one source, gate bars are drawn on both sources.
- Time gates can be placed anywhere. They can precede acquisition start value and extend beyond the acquisition value.
- The Time Gating bars are shown in both the overview and zoomed view when zoom is on.

See also

Measurement Name panel (Measurement configuration menu) (on page 300)

Configure panel (Measurement configuration menu) (on page 301)

Reference Levels panel (Measurement configuration menu) (on page 303)

Pass/Fail Testing panel (Measurement Settings menu) (on page 311)

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

Other measurement settings panels

Measurement Name panel (Measurement configuration menu) (on page 300)

Configure panel (Measurement configuration menu) (on page 301)

Reference Levels panel (Measurement configuration menu) (on page 303)

Clock Recovery panel (Measurement configuration menu) (on page 304)

Clock Recovery- Advanced Settings configuration menu (on page 308)

Gating panel (Measurement configuration menu) (on page 309)

Pass/Fail Testing panel (Measurement Settings menu) (on page 311)

Pass/Fail Testing panel (Measurement Settings menu)

Use these settings to test measurement values against a specified limit and set actions for the instrument to take on failure.

To open the **PASS/FAIL TESTING** panel:

1. Double-tap a Measurement badge.
2. Tap the **Pass/Fail Testing** panel.

Pass/Fail Testing 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	Description
Pass/Fail Test	Turns pass/fail testing on or off. Default is off.
Fail When	Sets the condition for when the test on the measurement will fail. Default is **< Limit**.
Limit	Sets the measurement limit value. This control is present when **Fail When** is not set to **Inside Range** or **Outside Range**.
Low Limit	Sets the low limit for the measurement value. This control is only present when Fail When is set to **Inside Range** or **Outside Range**.
High Limit	Sets the high limit for the measurement value. This control is only present when Fail When is set to **Inside Range** or **Outside Range**.
Actions on Failure	Opens a configuration menu to set the actions the instrument takes when the test fails.
Badge Navigation	Sets the navigation in the results badge. Tap to toggle to view **Failures** or **All Occurrences**. When Failure is selected the navigation buttons move from failure to failure. When All Occurrences is selected, the navigation button moves through each occurrence.

Other measurement settings panels

Measurement Name panel (Measurement configuration menu) (on page 300)

Configure panel (Measurement configuration menu) (on page 301)

Reference Levels panel (Measurement configuration menu) (on page 303)

Gating panel (Measurement configuration menu) (on page 309)

Menu bar overview

The Menu bar provides access to file, utility, and help functions.

The Menu bar

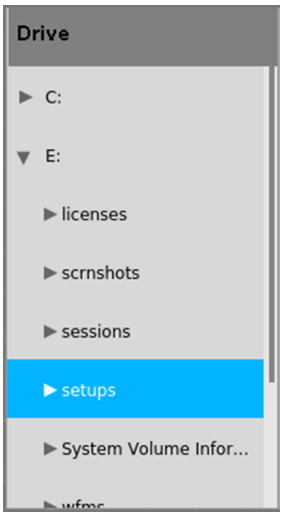
Field or control	Description
File	Provides typical system file management operations such as opening, saving, moving, and renaming files. See the links after this table for more information. See Recall configuration menu (File menu) (on page 313). **Save** saves current waveform, session, or setup file. See Save As configuration menu (File menu) (on page 314). See File Utilities configuration (File menu) (on page 319). **Default Setup** immediately restores the oscilloscope to factory default settings.

Field or control	Description
Edit	Provides a menu to Undo or Redo the last operation. See the links after this table for more information. See Undo, Redo (Edit menu) (on page 322).
Utility	Use to set user preferences, configure input, output, and network settings. See the links after this table for more information. See User Preferences (Utility menu) (on page 322).
Help	Opens the Help viewer, displays current instrument software, See the links below for more information. See Help... (Help menu) (on page 340). See About (Help menu) (on page 340).

Recall configuration menu (File menu)

Use this menu to recall (load) reference waveforms, instrument setups, mask settings, and sessions (setup plus waveforms). The supported file formats are .wfm, .trc, .isf, .tr0, .wfmx, .set, .tss, .msk, .xml, .bin, and .csv.

Recall configuration menu fields and controls

Field or control	Description
Look in:	Shows the current directory path to the location of a file. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, up to a maximum of 20 locations.
File type to open (tabs)	Tabs on the left let you set which type of file to recall (Waveform, Setup, Mask, 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.

Field or control	Description
File/Folders list view	List all files or folder for currently selected directory. List view displays more details about file, folder such as Name, Size, Modifed date and time, Type, etc. The image file formats supported for are .PNG, .JPG, and .BMP.
	Use the arrow buttons to navigate the file directory. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder. The Up arrow navigates up one level from the current folder.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File name	Lists the selected file name.
Files of type	Use to select the file format you want to open. The drop-down list shows all file extension types that the instrument can read for the selected file type.
Cancel	Cancels the file open action and closes the configuration menu.
Recall	Recalls the selected file. Recalling a waveform file adds a Reference waveform badge to the Settings bar and displays the waveform as it fits in the current Horizontal setting. Recalling a Setup or Session immediately sets the oscilloscope to the settings in the file and displays any waveforms that were part of the recalled session.

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.

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.

NOTE: Oscilloscope supports File Name with unicode characters for Screen Capture, Report, Mask, Setup, Session, and Waveform files.

Save As configuration menu fields and controls

The following fields and controls are common to all Save As actions.

Field or control	Description
File save type	Tabs on the left let you set which type of file to save (**Screen Capture**, **Waveform**, **Setup**, **Report**, or **Session**). Selecting a file type sets the file extensions in the Save As Type field to the correct value.
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Tap to open the **Browse Save As Location** configuration menu, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu (on page 318) .
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Auto Increment File Name	Select to enable auto-incrementing of the file name. Auto increment adds a number count to the end of the file name, and increments it on each subsequent saving of the same file name. You can enable auto-incrementing of the file name for saving files to/from local/network/USB/TekDrive file storage locations.
Count	Sets the increment count start number. Default is 000.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the save format.
Recall to Ref on Save	Select the checkbox to recall the saved reference waveform. Recall to Ref on Save is not supported when ▪ When saving files as MATLAB (.mat) or SignalVu (.tiq). ▪ When source are analog channel and FastAcq is enabled. ▪ Bus waveforms and when bus Source is set to All and one or more bus waveforms are present, only non-bus waveforms can be recalled as reference waveforms. ▪ Autoref recall is not applicable to the waveforms saved by Act on Event.
Cancel	Cancels the file save action and closes the configuration menu.

Field or control	Description
OK	Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.

Screen Capture tab fields and controls

The following settings are specific for saving a screen capture

Field or control	Description
File save type	Tap the **Screen Capture** tab to save a screen image to a file. Selecting Screen Capture sets the file extensions in the Save As Type field to available graphic file formats.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the graphic save format.

To save screen captures with waveforms on a white background, tap **Utility>User Preferences** and set **Screen Capture** Colors to **Inverted.** When you save the next screen capture, the display inverts colors for just a moment, saves the displayed image to the file, and returns the screen to normal colors. This capability us useful to save ink on printouts of screen captures. See the **Display** panel settings in User Preferences (Utility menu) (on page 322).

Waveform tab fields and controls

The following settings are specific for saving a waveform.

Field or control	Description
File save type	Tap the **Waveform** tab to save waveform(s) to a file. Selecting **Waveform** sets the file extensions in the **Save As Type** field to available waveform file formats.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the graphic save format.
Source	Sets the waveform source to save. You can save a single waveform, or save all active (displayed) waveforms. Waveform file names include the source channel or waveform appended to the end of the file name.
Display as next available Ref waveform	Sets the waveform to be saved and automatically opened (displayed) as the next available reference waveform. It doesn't matter what type of file is saved, that waveform will be opened as a Ref waveform. If gating is in use, a smaller, gated waveform is saved and displayed. This control is not present when source is set to **All**.

Field or control	Description
Gating	Sets the method to save a specified part of the waveform data. **None** saves the full waveform data (default). **Cursors** saves the waveform data located between the vertical cursors. If cursors aren't on when selecting cursor gating, the cursors are activated. **Screen** saves the waveform data that is on the screen. **Resample** saves the waveform data at a sample interval set by the user. The resulting saved waveform is a resampled version of the original waveform with fewer data points. Gating notes: ▪ Default Setup restores Gated Save to its default setting (None). ▪ The state of Gated Save gets saved in Setup and Session files. ▪ Gated saves do not work on plot waveform data. ▪ The **Cursor** gating type is not available when **Cursor Type** is set to **Custom**.
Clip Value Format	Specifies how clipped waveform samples are written when saving waveform data to a CSV file. Select **INF** to write clipped samples as the text value"INF". Select **Value** to write clipped samples as a numeric value one digitizing level beyond the displayed waveform range. Default is **INF**.

Setup tab fields and controls

The following settings are specific for saving an instrument setup.

Field or control	Description
File save type	Tap the **Setup** tab to save the instrument setup and measurement settings to a file. Selecting Setup sets the file extension in the Save As Type field to .set.
Include Reference Waveforms	Include the instrument waveforms.

Report tab fields and controls

The following settings are specific for saving an instrument report.

A report can include information on instrument settings, measurement results, screen images, individual measurement configuration, source input settings, and error and warning information for measurements. Reports are either a PDF file or a single file Web page.

Field or control	Description
File save type	Tap the **Report** tab to save a report file. Selecting Report sets the file extensions in the Save as Type field to available report file formats.
Save As Type	Lists the available formats to which you can save files. The available save formats are set by the type of file being saved. Tap the field and select the report save format (PDF or Single File Web Pages).

Field or control	Description
Append Report	Appends a report to an existing report file.
Comments	Add comments to clarify the contents or purpose of the report, or specifics of the signals being measured.
Include Images and Annotations	Add Waveform and Plot images to the report.
Include Setup Configuration	Add instrument and measurement configuration information to the report.

Session tab fields and controls

The following settings are specific for saving an instrument session.

A session is a zipped file that contains an instrument setup file and all acquired waveform data. Use session file contents to move analysis activities to a PC and free up the instrument for others to use.

Field or control	Description
File save type	Tap the **Session** tab to enable saving a session file. Selecting Session sets the file extensions in the Save As Type field to .tss.
Save As Type	A session file can only be saved to format .tss.

Browse Save As Location configuration menu

Use this menu to select a new file location to save files.

To access the Browse Save As Location configuration menu:

1. Select **File > Save As...**.
2. In the Save As configuration menu, tap the **Browse** button.

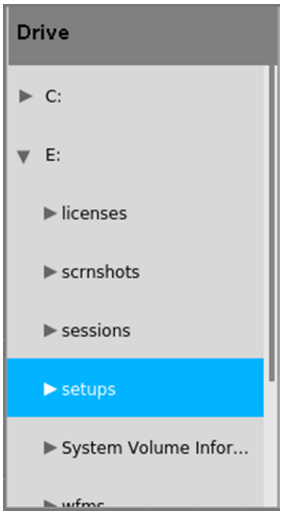
File operations and Microsoft Windows 10 Operating System SSD

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.

NOTE: Oscilloscope supports File Name with unicode characters for Screen Capture, Report, Mask, Setup, Session, and Waveform files.

Browse Save As Location configuration menu fields and controls

Field or control	Description
File path field	Shows the current directory. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recently accessed file save paths, for the current save type, up to a maximum of 20 paths.
	The Drive column lists the directory structure, opening at the root level. Use to quickly navigate to a location. Tap to list the contents of the directory in the Name pane. Double-tap an item, or tap the small arrow to the left of the directory, to display the subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
	Use the arrow buttons to navigate the file directory. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder. The Up arrow navigates up one level from the current folder.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
Cancel	Cancels the file path change, closes the configuration menu, and returns to the Save As configuration menu. No file path change occurs.
OK	Closes the configuration menu, returns to the Save As configuration menu, and updates the Save Location field with the path from the Browse configuration menu.

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 Utilities configuration menu fields and controls

Field or control	Description
File path field	Shows the current directory. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations, for the current save type, up to a maximum of 20 locations.
Drive navigation	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 sub-directories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
File/Folders list view	List all files or folder for currently selected directory. List view displays more details about file, folder such as Name, Size, Modifed date and time, Type, etc. The image file formats supported for are .PNG, .JPG, and .BMP. Double-tap the report file to open in a separate window. The supported formats are .MHT and .PDF. Use left/right arrow to navigate to the previous/next report.
	Use the arrow buttons to navigate the file directory. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder. The Up arrow navigates up one level from the current folder.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
Copy	Copies the selected file in the filename pane to memory. You can copy the Tek files or folders from local/network/USB drives or from TekDrive file storage location.
Paste	Pastes the file from the last Copy action in the current File Utilities session into the current location. You can paste the Tek files or folders to the chosen file target location. The target locations are local/network/USB drives or TekDrive file storage location as well.
Delete	Deletes the selected file or folder.
Rename	Renames the selected file or folder.
Unmount	Unmount the selected drive. Select the drive letter and tap **Unmount**. For network drives, **Unmount** removes the network drive connection information from the . The network drive is also removed from the Drive column of menus that can access drives. For TekDrive, **Unmount** removes the TekDrive connection information from the . The TekDrive is also removed from the Drive column of menus that can access drives.

Field or control	Description
Burn	Tap to burn oscilloscope files or folders to a CD or DVD. Refer to Burn files to a CD or DVD.

Mount Drive configuration menu

Use this menu to connect to a network or TekDrive.

To open the **Mount Drive** menu:

1. Select **File** > **File Utilities** from the Menu bar.
2. Tap **Mount** to open the **Mount Drive** menu.

Mount Drive configuration menu fields and controls

Field or control	Description
Drive Type	Select the required drive to mount on your . ▪ Network ▪ TekDrive
Domain/Workgroup	Sets the Domain/Workgroup of the intranet for the network drive.
Verbose	Tap the Verbose checkbox to get more information in the log file. Default is unchecked.
TekDrive	
Drive Name	Enter the required drive name for mounting.
Auto Disconnect	Select to configure the duration for the TekDrive connection to be active for usage. ▪ **PowerCycle**: Select to unmount the TekDrive after power cycling the . There is no time restriction when this option is chosen. ▪ **Never**: Select to mount the TekDrive connection permanently. ▪ **Custom**: Select to disconnect the TekDrive after a chosen duration. The default selected is **PowerCycle**.
Time	Enter the required time for the Auto Disconnect. The mounted TekDrives gets disconnected automatically from the after the specified time. Available only when **Auto Disconnect = Custom**.
Restrict connection to current IP Address	Select to enable the restriction of connection to the current IP address. This is performed for security purposes. Enabled checkbox setting is recommended. When the checkbox is disabled, the application displays the error: "This setting is recommended, because it restricts TekDrive access to the current public IP Address making your data secure and inaccessible if the device moves to a new network location."
Connect to TekDrive	Select to confirm all the configured settings and connect to the TekDrive.

Field or control	Description
Code activation controls	
Cancel	Select to close the menu without performing any action and does not connect to the new TekDrive.
Back	Select to take you to the previous menu giving an opportunity before connection, to reconfigure all the input control settings to connect to the TekDrive.
OK, Mount TekDrive	Select to confirm the configuration to complete the TekDrive mounting. This will make sure all the file browsers in the application show the connected TekDrives in it for saving or recalling files to TekDrive.

Undo, Redo (Edit menu)

The Edit menu lets you Undo or Redo . Tap Undo or Redo. Not all oscilloscope actions can be undone.

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
Language	Enables displaying the user interface and Embedded Help files in an available language. Tap the field and select the language. The user interface changes immediately to the specified language. New user interface elements or Help topics may not be translated at the release of updated firmware. The translations will be available in a future firmware release. The Help files support English, Japanese, and Simplified Chinese. All other language selections will display the Help topics in English. The information/warning messages display according to the selected language.
Font Size	Sets the font size for user interface text elements. Font size ranges from 12 to 20.

Field or control	Description
Default Waveform View Display Mode	Sets the default waveform view display mode, either **Overlay** or **Stacked**.
Waveform View Graticule	Sets the graticules to be **Movable** or **Fixed** for both stacked and overlay modes. **Movable** (default) sets graticules to move along with waveforms, and displays graticule units. **Fixed** sets graticules so that they do not move when you change a waveform's vertical or horizontal position. **Fixed** graticule mode also does not display the Vertical and Horizontal graticule labels. Use **Cursors** to take on-screen measurements.
Displayed Colors	Tap and select either **Normal** or **Inverted** colors to set how the instrument displays waveforms and plots. **Normal** shows waveforms and plots in color with a back background. **Inverted** makes the waveform background white, with graticule markings in black. **Inverted** does not change the Menu bar, Results bar, or Settings bar colors. Use this setting to save ink on printed screen captures.
Define Custom Colors	Opens the Define Custom Colors menu (on page 328). This menu lets you change channel, math, and reference waveform colors for both **Normal** and **Inverted** display modes.
Screen Capture Colors	Sets how the instrument saves screen images (**Normal** or **Inverted** colors). **Normal** saves waveforms and plots in the colors as shown on the screen. **Inverted** makes the waveform background white, with graticule marking in black, just for screen saves. **Inverted** does not change the Menu bar, Results bar, or Settings bar colors. Also, if **Displayed Colors** is set to **Inverted**, selecting **Inverted** for screen saves does not invert the colors back to normal for the screen save; to save normal colors for a screen save, select **Normal.**
Backlight	Sets the intensity of the backlight. **NOTE:** Performing an Autoset resets the backlight value to High.
Auto-Dim	Select **On** to automatically dim the screen backlight after a specified time.
Time	Sets the amount of time to wait before the display is dimmed. Tap in the field and use the knob to change the time value, or double-tap to open the virtual keypad and set a time value. Available when **Auto-Dim** is **On**.
Low Latency Trigger Out	When set to On, this control reduces the delay from when a trigger event is received to when the trigger is output to the AUX Out. This settings also restricts trigger type to Edge only and the trigger sources to Channel 1 or Aux In. When set to Off, delay is approximately 1 us. When set to On, delay is <20 ns.

Autoset panel fields and controls

Field or control	Description
Autoset Adjusts	Selects which controls to change as part of the Autoset operation (**Vertical Settings**, **Horizontal Settings**, **Trigger Settings**, and **Acquisition Settings**). The default is for all adjustments to be enabled.
Autoset in Overlay Display Mode Optimizes	Sets whether to optimize waveform resolution or visibility when doing an Autoset while in the Overlay display mode. **Resolution** 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. **Visibility** scales and positions waveforms so they are visually separated from each other at the expense of vertical resolution and measurement accuracy.

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
Shared Reference Levels	Tap to select either Global or Per Source reference levels. **Global** applies the same reference levels to all measurement sources. **Per Source** allows selection of a different reference levels for all measurement sources.
Calculate Reference Levels	Tap to select how often to calculate reference levels. **First Acquisition** only calculates the reference levels on the first acquisition and re-uses them for all subsequent acquisitions. Pushing the Clear front-panel button, or making measurement changes (adding, deleting, or reconfiguring) also clears the reference levels which are then recalculated on the next acquisition. **Every Acquisition** calculates the reference levels with every acquisition.
Measurement Interpolation	Interpolation defines how the waveform is drawn between waveform record data points. Tap and select the measurement interpolation method. **Auto** selects the best interpolation method. **Sin(x)/x** interpolation computes record points using a curve fit between the actual samples acquired. **Linear** interpolation computes record points between actual acquired samples by using a straight line fit.

Field or control	Description
Measurement Annotations	Annotations show the exact segments of the waveform from where the chosen measurements are taken. It is a visual aid to debug the signals. There are various markers which denote the measurement annotations such as horizontal bars, vertical bars, triangles (or carets), or arrows. **Auto** sets annotations to display if valid for the measurement. To view annotations for a measurement, select that measurement badge. If annotations are valid for that measurement, they are added to the waveform source for that measurement. **Off** turns off display of measurement annotations.

Measurement annotations reference

Supported measurements

- The advanced measurement annotations are shown for all the Amplitude and Time based measurements shown in Standard tab (in the Measurement Selection panel).
- Typically, Amplitude based measurements show the annotations with Top and Base horizontal lines/bars and a caret, signifying where the measurement occurred. Whereas the Time based measurements show reference levels with vertical lines/bars on the waveform cycle.
- On navigation, the annotations move back or forward (on per cycle basis) based on whether you tap Prev/Next buttons.

Visual elements

- An arrow annotation represents the value of the measurement reference level on the vertical bars.
- The caret type of annotation denotes the time position of the sample point at that location in a waveform record. It is also shown in conjunction with the dotted line pointing towards it, running across the waveform horizontally.
- The vertical lines denotes the start and end of the cycle and the waveform record.
- The horizontal dotted lines correspond to the voltage levels found by the base-top method.

Miscellaneous

- The measurement annotations are associated with only analog source waveforms (Ch/Math/Ref).
- The setup/session is saved with active measurements and its annotations together. When the setup/session is recalled, it also recalls the user preference settings.

Measurement limits

- The annotations are displayed on the filtered result collection within the limits.
- The limit showing measurement annotations for the measurement result values to those that are within the specified Min - Max range.
- If there are no measurements in the specified range then annotations are disabled.
- If limits are off then the annotations are displayed for the entire measurement result collection.

Measurement Pass/Fail Testing

- The annotations will be shown on the filtered result collection within the pass/fail test limits.
- If pass/fail testing is on, it limits showing the measurement annotations for the measurement result values that pass the pass/fail test criteria.
- If pass/fail testing is off, the measurement annotations are displayed for all the edges in the acquired waveform record.

- If pass/fail testing is on and badge navigation is set as failures, the annotations will be limited to only for the failed measurement result values.
- If there are no failures found in pass/fail testing, then annotations are disabled.
- If pass/fail is on and badge navigation is set for all occurrences, then the annotations are displayed for all measurement edges/values (irrespective whether it passes/fails).

Measurement Gating

Annotations work as expected for all the gating types, except for Search and Logic gating.

Jitter and Eye Analysis panel fields and controls

Field or control	Description
Calculate Reference Levels	Tap to select calculating the reference levels on the first acquisition or on every acquisition.
Horizontal Measurement Units	Tap to select horizontal measurement units of seconds or unit intervals.
Jitter Separation Model	Tap to select the jitter separation model (**Spectral Only** or **Spectral + BUJ**).
Minimum # of UI for BUJ Analysis	Enter the number of unit intervals for BUJ. Available when **Jitter Separation Model** is **Spectral + BUJ**.
Lock RJ Value	Selecting **Lock RJ Value** calculates the measurements at the specified random jitter value. The checkbox is unchecked by default. Selecting the checkbox displays a text box where you can enter the RJ value. The default value is 1 ps. **NOTE:** Lock RJ Value cannot be configured when **Jitter Separation Model** is **Spectral + BUJ** or when **Jitter Analysis Method** is set to **Jitter + Noise.**
DDJ Separation Model	Tap to select **Spectral Domain** or **Time Domain**. **Spectral Domain** approach measures the DDJ for waveforms using a spectral approach. **Time Domain** approach the DDJ value is calculated in Time domain as max{DDJn} – min{DDJn} for a waveform with a repeating pattern having N edges per pattern cycle. Where DDJn is the DDJ-per-edge calculated from the time-domain TIE trend. The DDJ values calculated by the spectral-domain and time-domain methods converge as record length grows. For short waveforms where the number of observations per pattern edge is low, small variations may be observed between the measurement methods. **NOTE:** The DDJ Separation Model option is Available only when Jitter Analysis Method is set to **Jitter Only.**
Enable Jitter Measurement Based on N-Unit Interval	Enable this checkbox to perform Jitter Measurements Based on Period or N-Unit Interval. When this is enabled a new configuration appears in applicable jitter measurements for choosing the Base Measurement Technique. **NOTE:** The Enable Jitter Measurements Based on N-Unit Interval option is Available only when Jitter Analysis Method is set to **Jitter Only.**

Field or control	Description
Jitter Analysis Method	Specifies the method used for jitter and noise decomposition. **Jitter Only**: In this mode, only the Jitter Components are decomposed. **Jitter And Noise**: In this mode, both Jitter and Noise Components are decomposed. **NOTE:** Noise measurements will give results only in Jitter And Noise mode. See Section Jitter Analysis Method (on page 595) for more details.

Other panel fields and controls

Field or control	Description
Right Clicks via Touch	Turns **On** or **Off** the ability to use touch and hold method to open right-click menus on badges and other screen items.
Time	Sets the time it takes to respond to a touch and hold before opening a right-click menu.
Low Latency Trigger Out	Sets the latency or time taken by an event to propagate from the input channel and/or Aux In on the front of the oscilloscope to the Aux Out BNC on the rear of the oscilloscope. When enabled, it notifies in the Trigger menu. This latency can often exceed 1 µS. A low-latency trigger mode for Channel 1 and/or Aux In which has a delay less < 50 ns.
Programmatic Interface Backward Compatibility	Allows you to choose an XML file that ensures backward compatibility of the commands defined in the XML file. Toggle button turns **On** or **Off** the backward compatibility option. Default is off. For more information refer to *7 Series DPO Programmer Manual*.
Load	Tap to select the XML file. You can navigate to TekDrive to load XML file. Appears when **Programmatic Interface Backward Compatibility** is **On**.
Connected Scope Preferences	Enables the connected oscilloscope with additional capabilities and improves the overall experience while using the oscilloscope. **NOTE:** The oscilloscope should be connected to the internet.
Oscilloscope tracks feature usage to help reproduce and fix software bugs. User identifiable information, waveform data, analysis results or screenshots are never saved	Enables oscilloscope tracks feature usage to help reproduce and fix software bugs. It will not save user-identifiable information, waveform data, analysis results or screen shots.

Connected Oscilloscope preferences

Use the Connected Oscilloscope preferences to make you more efficient and improve the overall user experience while using the oscilloscope ().

When enabled it provides additional capabilities which improves the overall experience while using the oscilloscope. Whenever the status of checkbox changes it is shared with Tektronix.

NOTE: Enabling this checkbox does not send any information that is used to identify a specific company or individual.

Tektronix will not send any information to the oscilloscope unless you enable Oscilloscope tracks feature usage to help reproduce and fix software bugs. User identifiable information, waveform data, analysis results or screenshots are never saved.

When this option is enabled, the oscilloscope will record information about the instrument and its usage. The information includes actions taken in the user or programming interfaces, the model number of the scope, its firmware version, and information about installed options or probe attachments. This information does not include anything that is used to identify a specific company or individual.

This information is stored in a text file on the instrument and is viewed with any text viewing program. Disabling this feature will delete the file. If the instrument is connected to a network while this feature is enabled, this information will be periodically sent to Tektronix.

By enabling this option you agree that the software records anonymous information about the usage of this instrument, and periodically send it to Tektronix.

Tap **Legalese** to view legal information within the legal framework.

Tap **Save Preferences** will save the selected preferences.

Tap **Cancel** to discard the changes.

Legalese

Tap the Legalese button to view the legal information and details of the functionalities enabled when you check the boxes in the previous dialog (Connected Preferences).

To open the Legalese dialog, select

When the check box is enabled in dialog: You agree that the software may contact Tektronix servers periodically. You also agree that the instruments model and serial number and status of the enabled check boxes will be sent to Tektronix.

When the check box is enabled in dialog: You agree that the software may record anonymous usage information about this instrument and send it to Tektronix. You also agree that the instruments model and serial number and status of the enabled check boxes will be sent to Tektronix.

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
Normal Display Colors	Use this menu to change the waveform colors to use when the instrument is in Normal display mode. **Channels** sets the colors used to display instrument channels. **Maths** sets the colors used to display math waveforms. **Refs** sets the colors used to display reference waveforms. 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 **Normal** mode. The color menu stays open to let you experiment with different waveform colors. Tap outside the color menu to close it.
Inverted Display Colors	Use this menu to change the waveform colors to use when the instrument is in Inverted display mode. **Channels** sets the colors used to display instrument channels. **Maths** sets the colors used to display math waveforms. **Refs** sets the colors used to display reference waveforms. 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 **Inverted** mode. The color menu stays open to let you experiment with different waveform colors. Tap outside the color menu to close it. The colors for Math waveforms 1-8 are reused for Math waveforms 9-16, 17-24, and so on. The colors for Ref waveforms 1-8 are reused for Ref waveforms 9-16, 17-24, and so on.
Set All Normal Display Colors to Default	Sets all **Normal** display mode waveform colors to their factory default settings.
Set All Inverted Display Colors to Default	Sets all **Inverted** 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 or LAN SFP+ panel, the Network Address is set to **Auto** (default setting) and the **Apply Changes** button is grayed out (inactive) in the LAN panel.

When you select any editable input box and start entering data, the **Apply Changes** button becomes active, and the characters being entered are bolded and italicized. Bold italicized text means that the values have not been applied to the oscilloscope settings.

When you tap the **Apply Changes** button, all changes are saved (takes about 10 seconds), the text is changed to normal font (nonbold, nonitalic), and the **Apply Changes** button becomes inactive.

If you tap outside the I/O menu before you tap the Apply Changes button, the menu closes and none of your changes are saved.

LAN panel fields and controls

Field or control	Description
LAN Status	A readout that indicates the status of the LAN connection, either a Green circle with the word Normal or a Red circle with an error message.
Host Name	The instrument host name is displayed. To change the name, double-tap and enter a name in the virtual keyboard.
Network Address	Tap to select **Manual** or **Automatic** mode. In Automatic mode, the current Instrument IP Address, Gateway IP Address, Subnet Mask, and DNS IP Address are displayed.
Domain Name	The instrument domain name is displayed. To change the name, double-tap and enter a name in the virtual keyboard.
Instrument IP Address	Tap and use the multipurpose knobs to enter the address. Use the A knob to select the digit, and the B knob to change the value. Available to edit when **Network Address** is **Manual**.
Subnet Mask	Tap and use the multipurpose knobs to enter the mask. Use the A knob to select the digit, and the B knob to change the value. Available to edit when **Network Address** is **Manual**.
Service Name	The instrument service name is displayed. To change the name, double-tap and enter a name in the virtual keyboard.
Gateway IP Address	Tap and use the multipurpose knobs to enter the address. Use the A knob to select the digit, and the B knob to change the value. Available to edit when **Network Address** is **Manual**.
DNS IP Address	Tap and use the multipurpose knobs to enter the address. Use the A knob to select the digit, and the B knob to change the value. Available to edit when **Network Address** is **Manual**.

Field or control	Description
MAC Address	A readout of the instrument MAC Address.
Test Connection	Tap to test the connection. If the connection test is successful, then **OK** is displayed. If the test is unsuccessful, then **No Response** is displayed.
LAN Reset	Tap to display the LAN and LAN SFP+ Reset configuration menu (Utility > I/O menu) (on page 334).
Apply Changes	Apply changes made on this panel to the instrument. No changes are made to instrument settings until you tap the **Apply Changes** button.

USB Device Port fields and controls

Use the USB Device Port panel to enable or disable USB ports. Use USB ports to connect a USB memory device, keyboard, or for direct PC control of the oscilloscope using USBTMC protocol.

Field or control	Description
USB Device Port	Tap to toggle all USB device ports **On** or **Off**.
USBTMC Configuration	Displays the USBTMC configuration information.

Socket Server panel fields or controls

Use the following settings to setup and use a socket server between your oscilloscope and a remote terminal or computer.

Field or control	Description
Socket Server	Tap to toggle the server **On** or **Off**.
Protocol	Tap to select a protocol, either **None** or **Terminal**. A communication session run by a user at a keyboard typically uses a terminal protocol. An automated session might handle its own communications without such protocol from the oscilloscope.
Port	Enter the port number using the multipurpose knob or virtual keypad.

VXI-11 Server panel fields or controls

Use the following settings to setup and use a VXI-11 server between your oscilloscope and a remote terminal or computer.

Field or control	Description
VXI-11 Server	Tap to toggle the server **On** or **Off**.
Port Low	Enter the port low number using the multipurpose knob or virtual keypad.
Port High	Enter the port high number using the multipurpose knob or virtual keypad.

AUX Out panel fields and controls

Use the following settings to select the signal that is output on the rear-panel AUX Out signal connector.

Field or control	Description
AUX Out Signal	Tap to select the signal type to send to the AUX Out connector. **Trigger** sends a pulse for each trigger occurrence. **AFG Sync** sends a pulse that is synchronized to the AFG output signal.
Polarity	Tap to select the polarity of the trigger signal (positive or negative pulse per trigger event). The polarity control is present only 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 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 lower menu button and the resulting LAN Settings side menu button to view the resulting Ethernet 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 backspace key on the keyboard during an Windows Telnet session with the oscilloscope.

High Speed Interface panel fields and controls

The High Speed Interface settings are available only when both the LAN Port and Remote Access Security Options are enabled.

Field or control	Description
High Speed Interface	Tap to toggle the High Speed Interface **On** or **Off**. Default is **On**.
Port	Enter the port number using virtual keypad. Default value is 5000.
Timeout	Sets the time period for the High Speed Interface to timeout. Default value is 20 s.
Username	Displays the username used for authenticated High Speed Interface (HSI) connections. Default value is **Tektronix** (cannot be changed). Use the password configured in **Utility > Security > Network-enabled remote access** for authentication.

Field or control	Description
Encryption	Tap to toggle the Encryption **On** or **Off** for High Speed Interface communications. Default is **On**. In combination with "Encryption" and "Password-protect this setting" for **Utility > Security > Network-enabled remote access**, the following three modes are available for HSI communication. **Mode 1**: No Encryption / No Authentication (Isolated Lab Only) Selected when Utility>Security>Network-enabled remote access>Password-protect this setting = Unchecked, Encryption = Off. ▪ The TekHSI server accepts plaintext connections without authentication. ▪ This mode is intended for isolated lab networks only and must not be the recommended default for customers on shared networks. ▪ TLS-only clients will not be able to connect; this is by design. **Mode 2**: Encryption Only (TLS, TOFU) Selected when Utility->Security->Network-enabled remote access->Password-protect this setting = Unchecked, Encryption = On. ▪ The TekHSI server serves TLS using its built-in self-signed certificate. ▪ No credential is required to connect. ▪ Clients use the TOFU model: on the first connection they accept and pin the server's certificate; on subsequent connections they compare against the pinned fingerprint. ▪ If the pinned fingerprint and the presented certificate do not match, the client must surface a clear identity-mismatch error and must not silently proceed. **Mode 3**: TLS + Username and Password Selected when Utility->Security->Network-enabled remote access->Password-protect this setting = Checked, Encryption = On (Automatically Turned On). ▪ The TekHSI server requires TLS and valid credentials (username + password) on every connection. ▪ Username defaults to "Tektronix". ▪ When Password is turned On, Encryption is automatically turned On and becomes non-editable. ▪ Invalid credentials result in connection rejection and a logged authentication failure. **NOTE:** • **TLS** - Transport Layer Security • **TOFU** - Trust On First Use is a security model in which a client implicitly trusts a server, device, or key the first time it connects, and then remembers that identity for future connections.

For more information on High Speed Interface, refer tek.com/solutions/application/test-automation/instrument-high-speed-interface.

LAN and LAN SFP+ 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

See also

I/O (Utility menu) (on page 329)

Self Test configuration menu (Utility menu)

Use this menu to view power-on diagnostic results and run extended self tests.

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
Export Log Files	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. You can navigate to the TekDrive location to export log files.
Power-on Self test Results	Displays the power-on self test status (**Passed** or **Failed**).
Extended Self Test Results	Lists the status of each Extended self tests (**Passed** or **Failed**). If one or more tests fail at power-on, tap on **Run Self Test** 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.
Run N Times	Sets the number of times to run the extended self tests.

Field or control	Description
Run Self Test / Abort Self Test	Tap **Run Self Test** to run the extended self tests. While tests are running, the button changes to **Abort Self Test**. When self tests are stopped, the button reverts to **Run Self Test**. **NOTE:** Remove all input signals before running the extended self tests. Tap the **Abort Self Test** 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
Factory Adjustment Status	This area at the top of the menu lists the instrument calibration status. Factory Adjustment Status should be Calibrated. If an instrument becomes uncalibrated, this displays an Uncalibrated status and a red Warning message bar is shown at the top of the screen in the Menu bar area. Contact your nearest Tektronix Service Center for assistance.
SPC Status	Indicates the status of the last SPC run (Pass or Failed). Also indicates how long ago the last SPC was run.
Run SPC	Signal path Compensation (SPC) corrects for internal DC inaccuracies caused by temperature variations and/or long-term drift in circuits. **NOTE:** SPC takes 15 minutes per channel to run. Allow the instrument to warm up for 20 minutes before running SPC. Remove all probes, cables, and adapters from the input connectors before running SPC. Tap Run SPC to run signal path compensation. The compensation can take up to 15 minutes per channel.

Security configuration menu

Use this menu to configure Network-enabled remote access, LAN port and securely erase the oscilloscope nonvolatile memory (TekSecure®). Access to the functions on these menus are password protected.

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 **Network-enabled remote access** fields.
4. Tap **Password-protect this setting** or **Password-protect these settings** check box to access the remote access or oscilloscope security option menu. The check box is unchecked by default.

NOTE: When **Password-protect this setting** is checked, the screen displays Enter Password (on page 337) menu before any changes to the settings can be made. You will be asked for a password once each time you open the Security configuration menu.

Security configuration menu fields and controls

Field or control	Description
Remote Access Security Option	
Network-enabled remote access	Tap to toggle the remote network access **On** or **Off**. Remote access controls the following connectivity features; VXI-11, , and Socket Server. Each of the connectivity features can also be toggled On and Off individually in the corresponding I/O (Utitlity) menu panel.
Password-protect this setting	Opens the Set Password configuration menu (on page 338) to enable the password protection for the remote access.
Change Password	Opens the Change Password configuration menu (on page 338). This menu option is available after a password has been set.
Oscilloscope Security Options	
LAN Port	Tap to toggle the Ethernet port **On** or **Off**.
LAN SFP+ Port	Tap to toggle the Ethernet port **On** or **Off**.
USB Device Port	Tap to toggle the USB device port **On** or **Off**.
Password-protect these settings	Opens the Set Password configuration menu (on page 338) to enable the password protection for the oscilloscope security options.
Change Password	Opens the Change Password configuration menu (on page 338). This menu option is available after a password has been set.
OK	Tap to save the configured settings. The OK button is only available the first time you configure the Security menu settings.
TekSecure Erase Memory	Tap **Run TekSecure** to use the TekSecure Erase Memory (on page 339) function to erase nonvolatile memory.

Enter Password configuration menu

Use this function to enter the password to access optional security functions.

To enter the password to allow changing the selected security option (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 security option changes are saved and the menu closes.

Enter Password configuration menu fields and controls

Field or control	Description
Password	Enter the password. If you cannot use a keyboard because the USB ports are disabled, double-tap on the password fields to open the virtual keyboard and enter the password.
OK	Closes the menu and toggles the selected option when the entered password is correct. Any other conditions result in an error message.
Cancel	Closes the menu without taking any action.

CAUTION: 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 (on page 338).

Set Password configuration menu

Use this function to set the password used to access the optional security functions.

To access the Set Password configuration menu:

1. Tap **Utility** in the Menu bar.
2. Tap **Security**.
3. Tap **Password-protect this setting**.

Set Password menu fields and controls

Field or control	Description
New Password	Enter the new password. If you cannot use a keyboard because the USB ports are disabled, double-tap on the password fields to open the virtual keyboard and enter the password.
Confirm New Password	Re-enter the new password.
OK	Sets the new password and closes the menu when the new password is entered and confirmed. Any other conditions result in an error message.
Cancel	Closes the menu without taking any action.

CAUTION: If you need to send the instrument for service or repair, make sure to use the Delete Password (on page 338) 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

Use this function to change an existing password used to access the optional security functions.

To access the Change Password configuration menu:

1. Tap **Utility** in the Menu bar.
2. Tap **Security**.
3. Tap **Change Password**.

Change Password menu fields and controls

Field or control	Description
Current Password	Enter the current password. If you cannot use a keyboard because the USB ports are disabled, double-tap on the password fields to open the virtual keyboard and enter the password.
New Password	Enter the new password.
Confirm New Password	Re-enter the new password.
OK	Changes the password and closes the menu when the new password is entered and confirmed. Any other conditions result in an error message.
Cancel	Closes the menu without taking any action.

CAUTION: If you need to send the instrument for service or repair, make sure to use the **Delete Password** function before sending to the Tektronix Service Center. Failure to do so may impede servicing the instrument.

TekSecure Erase Memory

Use this menu to use the TekSecure™ function to erase the oscilloscope nonvolatile memory.

TekSecure is a shortcut for overwriting the SSD from the recovery partition.

•

TekSecure will also remove the password protection in the Security configuration menu.

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. TekSecure runs for and cannot be stopped once the process has started. Do not power down the instrument while TekSecure is running.
4. To exit the dialog without running TekSecure, tap outside of the configuration menu.

   TekSecure does not erase calibration constants or instrument firmware. Save any important waveform, screen capture, instrument setup, report, and session files to external memory before running TekSecure.

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	The upper half of the menu shows an overview of the summary and procedures for the selected demonstration button of the available tabs. The available tabs are **Misc** and **Serial**.
Connection Details	Tap this button to open a dialog box that shows how to connect the oscilloscope to a signal source to perform the selected demonstration.

Field or control	Description
Recall Demo Session	Tap this button to load the session file for the selected demonstration.
Demo tabs	Each tab consists of a number of demonstration buttons of the selected category. The available categories are **Miscellaneous** and **Serial Bus** within the selected tabs. 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.

License Options...(Help menu)

Use the License Options menu to view the list of possible license options along with the description about the licenses that can be installed in the oscilloscope. This aids you to select the right software licenses before purchasing.

To access this option, select **License Options** from the Help menu. **License Options** dialog is displayed that contains the possible licenses information. Tap on any tab and select a license to know about the selected license.

Select an option to view the related information about the selected option on top, next to the illustration. The related nomenclature and the recommended firmware version required is also displayed.

A small green tick appears when an option is already installed/enabled in the instrument.

About (Help menu)

Use the About configuration menu to show instrument information and installed options, and to install or uninstall analysis or feature options.

To open the About menu:

1. Tap **Help** on the menu bar.
2. Select **About** from the menu to open the About configuration menu.

About configuration menu fields and controls

Field or control	Description
System information	Provides system-related information such as model, bandwidth, serial number, Host ID, and installed firmware version. Provide this information when communicating with Tektronix to purchase option licenses or communicate with Customer Support.
Installed Options	Lists installed options. Click on an item in the list to show details of the option in the Option Details area below the list. See Install option upgrade licenses (on page 30).
Option Details	Lists details of the option selected in the list, including license type, when the license was checked out, when it was installed on the instrument, and the date the license expires.
Remove License	Opens the Location to Save the Exit Key dialog. Use this to uninstall a floating license option for use by others. You can use TekDrive to save removed license files. See Location to Save Exit Key configuration menu (on page 341).

Field or control	Description
Install License	Tap this button to open the Browse License Files dialog to navigate to and select an option license file to install. You can navigate to the TekDrive location to install saved license files. See Browse License Files menu (Help > About) (on page 342).

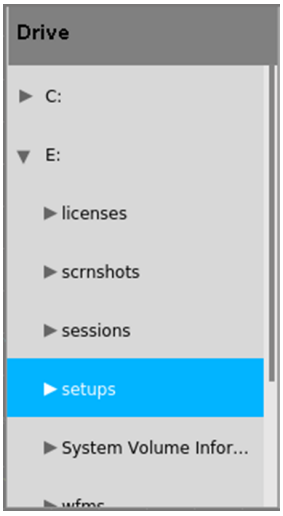
Location to Save Exit Key configuration menu

Use this menu to navigate to and set the location to save the option license key when you uninstall the license.

To access the Location to Save Exit Key menu:

1. Tap **Help** on the menu bar.
2. Select **About** from the menu.
3. Tap a license in the list that you want to uninstall (return the license).
4. Tap the **Remove License** button to open the Location to Save Exit Key configuration menu.

Location to Save Exit Key configuration menu fields and controls

Field or control	Description
Look in	Shows the current directory path at which to save the exit key file. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recently accessed paths, up to a maximum of 20.
	The Drive column lists the directory structure, opening at the root level. Installed USB memory devices are listed. Use to quickly navigate to a location. Tap to list the contents of the directory in the Files pane. Double-tap an item, or tap the small arrow to the left of the directory, to display the subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries. You can also use the files Name column to navigate to and select a folder in which to save the file.
	Use the arrow buttons to navigate the directory structure. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder. The Up arrow navigates up one level from the current folder.

Field or control	Description
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File Name	Lists the selected license file name to return (uninstall). Tap the down arrow icon on the right end of the field to open a list of uninstalled license files, up to a maximum of 20.
Files of Type	Use to select the file format you want to open. Tap the field to show a list of all file extension types that the instrument can read for the selected file type. The Name field (where folder and files are listed) only lists files of the specified type.
Cancel	Cancels the license uninstall process, closes the configuration menu without saving any changes that were made, and returns to the About configuration menu.
Create	Closes the configuration menu and saves the license information to the specified location.

Browse License Files menu (Help > About)

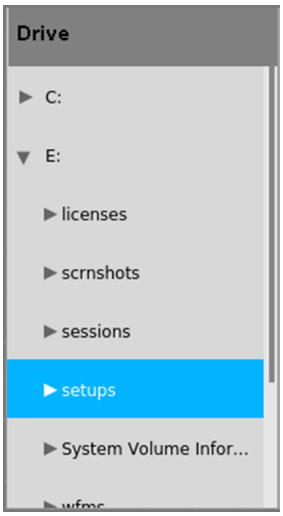
Use this menu to select and install an option license file to enable new functions.

To access the Browse License Files menu:

1. Tap **Help > About...** on the menu bar.
2. Tap **Install License** button to open the Browse License Files configuration menu.

Browse License Files configuration menu fields and controls

Field or control	Description
Look in	Shows the current directory path and file name. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.

Field or control	Description
	The Drive column lists the directory structure, opening at the root level. Installed USB memory devices are listed. Use to quickly navigate to a location. Tap to list the contents of the directory in the files listing. Double-tap an item, or tap the small arrow to the left of the directory, to display the subdirectories under it. Double-tap again to close that directory structure. Drag the list up and down to show more entries.
	Use the arrow buttons to navigate the directory structure. The left arrow navigates back to the previously visited folder. The Right arrow navigates forward to the previously visited folder.
	Use to create a new directory (folder) at the current location. Opens the new directory after it is created.
File Name	Lists the selected file name. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the on-screen keyboard and enter the new name. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.
Files of Type	Use to select the file format you want to open. Tap the field to show a list of all file extension types that the instrument can read for the selected file type. The files pane only lists files of the specified type.
Cancel	Cancels the configuration menu changes, closes the menu, and returns to the prior menu without making any changes.
Open	Closes the configuration menu, returns to the About configuration menu, and installs the license. Follow any instructions that may be shown during the installation.

More (DRAW A BOX Menu)

Use this menu to toggle the Waveform view box draw mode between Zoom area mode and Mask segment mode.

To use the **More (DRAW A BOX)** menu:

1. the **More** button (located at the top right of the Results Bar) to open the **DRAW A BOX** window.
2. Tap **Zoom** on the **DRAW A BOX** window to enable drawing a zoom box area on the screen. The mode stays in Zoom mode until changed.
3. Tap **Mask** on the **DRAW A BOX** window to enable drawing Mask Testing segments on the screen. The mode stays in **Mask** mode until changed.
4. Tap outside the menu. The changes to reflect the function .

See also

Turn on Zoom mode (on page 110)

Visual Trigger concepts (on page 542)

Create Visual Trigger areas (on page 544)

Edit visual trigger areas on the screen (on page 544)

Edit visual trigger areas using the Area menu (on page 546)

Add Plot configuration menu

You can 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. You can also plot a TIE measurement eye diagram or a Transfer Function between measurements.

Add Plot menu fields and controls

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

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.

Acq (Acquisition) Trend plot configuration menu

Use the Acq Trend configuration menu to set the acq trend plot waveform vertical scale and position, as well as add a label.

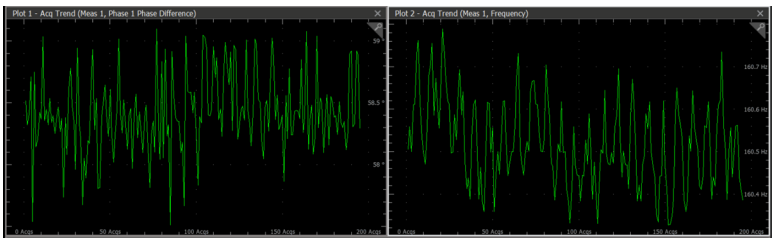
To open the Acq Trend configuration menu, double-tap the **Acq Trend** badge in the Measurement badge. You can also double-tap the Acq Trend Plot handle.

NOTE: The Acq Trend plot is available for Power Quality, Efficiency, Speed, and Direction measurements.

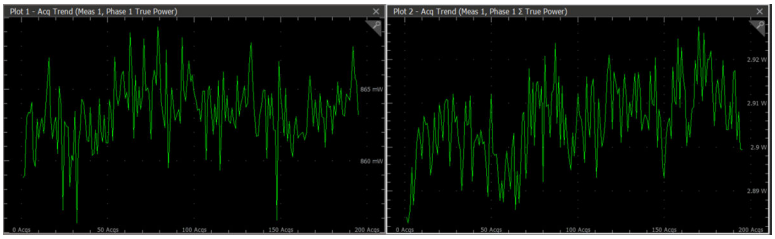
Acq Trend represents plotting of a selected measurement for each acquisition on a separate plot.

NOTE: Measurements that can use the Acq trend plot as a source will list the plot in the Source menu list.

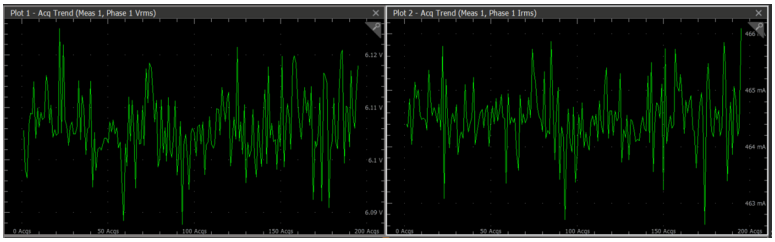
Acq trend plot resets when scope acquisition resets because of change in scope settings and measurement sources.



Phase Frequency



True Power



Vrms and Irms

Acq Trend configuration menu fields and controls

Field or control	Description
Auto Scale	Enables or disables the Auto Scale mode. Auto Scale uses the signal data to dynamically set the vertical scale units.
Plot Display	Available values are ▪ Phase 1(V, I) ▪ Phase 2(V, I) ▪ Phase 3(V, I)

Field or control	Description
Measurements	Available measurements for Power Quality are: ▪ Vrms ▪ Irms ▪ Phase Difference ▪ Frequency ▪ True Power ▪ Apparent Power ▪ Reactive Power ▪ Sum of True Power ▪ Sum of Apparent Power ▪ Sum of Reactive Power Available measurements for Efficiency are: ▪ Input Power ▪ Output Power ▪ Efficiency ▪ Total Efficiency ▪ Sum of Input Power ▪ Sum of Output Power Available measurements for Speed are: ▪ Speed ▪ Acceleration Available measurements for Harmonics are: ▪ THD-F ▪ THD-R
Save	Saves the plot data along with time stamp at the specified location.

Deleting a trend plot

To delete a Acq Trend plot, touch and hold the **Acq Trend** badge to open the right-click menu, and select **Delete Acq Trend**. Deleting the Measurement badge that enabled the plot also closes the plot.

Bathtub plot configuration menu

Use this menu to change settings of a displayed jitter versus BER (bathtub) plot.

To open the Bathtub plot configuration menu, double-tap anywhere in the **Bathtub** plot view.

To close a plot view, tap the X in the upper right corner of the view. Deleting the Measurement badge that opened the plot also closes the plot.

Settings panel (Bathtub plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Units	Sets the X-axis units to be unit intervals or seconds.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in BER units. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis scale	Sets the Y axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

Save panel fields and controls

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

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

Closing a plot view

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

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

Composite Jitter Histogram (CJH) plot configuration menu

Composite Jitter Histogram (CJH) plots four histograms (TJ, PJ, RJ+BUJ, DDJ) together. It is helpful in analysing the four components of jitter together. This plot is applicable for Jitter measurements only.

Use this menu to change settings of CJH plot.

To open the plot configuration menu, double-tap anywhere in the **CJH** 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 (Composite Jitter Histogram plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Num of Bins	Number of bins used in creating the histogram
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis scale	Sets the Y axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

Jitter Components fields and Controls

Use this panel to control the Jitter component Histograms displayed on the CJH Plot. A checkbox corresponding to the four histograms are provided.

Field or control	Description
TJ	Select TJ checkbox to display TJ histogram.
RJ+NPJ	Select RJ+NPJ checkbox to display RJ+NPJ histogram.
PJ	Select PJ checkbox to display PJ histogram.
DDJ	Select DDJ checkbox to display DDJ histogram.

Save panel fields and controls

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

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

Closing a plot view

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

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

Composite Noise Histogram (CNH) plot configuration menu

Composite Noise Histogram (CNH) plots five histograms (TN, PN, RN+NPN, DDN(0) and DDN(1)) together. It is helpful in analysing the five components of noise together. This plot is only applicable for Noise measurements.

Use this menu to change settings of CNH plot.

To open the CNH plot configuration menu, double-tap anywhere in the **CNH** 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 (CNH plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are **Horizontal**, **Vertical**, and **Both**.
Y-Axis Num of Bins	Number of bins used in creating the histogram
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is **Off**.

X-Axis scale	Sets the X axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

Noise Components fields and Controls

Use this panel to control the Noise component Histograms displayed on the CNH Plot. A checkbox corresponding to the five histograms are provided.

Field or control	Description
TN	Select TN checkbox to display TN histogram.
RN+NPN	Select RN+NPN checkbox to display RN+NPN histogram.
PN	Select PN checkbox to display PN histogram.
DDN(0)	Select DDN(0) checkbox to display DDN(0) histogram.
DDN(1)	Select DDN(1) checkbox to display DDN(1) histogram.

Save panel fields and controls

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

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

Closing a plot view

To close (delete) a Plot view, tap the X in the upper right corner of the view. Deleting the Measurement badge that enabled the plot also closes the plot.

Noise Bathtub plot configuration menu

This plot shows the extrapolated curves due to noise like the way Bathtub plot shows the extrapolated curve due to jitter.

Noise Bathtub Plot Configuration Menu

Use this menu to change settings of Noise Bathtub plot.

To open the Noise Bathtub plot configuration menu, double-tap anywhere in the **Noise 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 (Noise Bathtub plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Units	Sets the X-axis units to be unit intervals or seconds.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in BER units. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis scale	Sets the Y axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

Save panel fields and controls

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

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

Closing a plot view

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

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

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 344).

Settings panel (Eye Diagram plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles **AutoScale** on or off. Turn AutoScale off to manually set the X and Y-axis range to view an area of interest. When **AutoScale** is **Off**, a small Zoom window appears in the plot. Drag the blue vertical bar in the **Zoom** window to view that area in the main Plot view. Use the **Zoom** and **Pan** front-panel knobs to change the zoom area and position.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. Only available when **AutoScale** is **Off** (unchecked).
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. Only available when **AutoScale** is **Off** (unchecked).
Gridlines	Selects which gridlines to show in the plot. Select the grid style that meets your measurement needs.
Eye Rendering	Specifies the eye rendering as **Fast**, **Complete Vectors**, and **Complete Dots**. Fast eye rendering shows the UIs (Unit Intervals) that define the boundaries of the eye along with a specified number of surrounding UIs for added visual context. Complete Vectors shows all valid UIs, and the samples are interpolated to produce smoother, high resolution rendering. Complete Dots renders all samples of the valid UIs without performing any interpolation.
Bit Type	Sets the bit type as **All, Transition**, and **Non-Transition**.
UIs	The user specified number of surrounding UIs for added visual context in Fast rendering. Available only when **Eye Rendering** is **Fast**.

Field or control	Description
Superimpose Explicit Clock	When checked, overlays the explicit clock waveform on the data eye diagram, allowing direct visual comparison of clock and data timing relationships. This control is only available when the measurement source is configured with an explicit clock.
Vertical Scale To	Sets the vertical scale reference for the superimposed display. Available options are **Data** and **Explicit Clock**. Data scales the vertical axis to the data signal amplitude. Explicit Clock scales the vertical axis to the clock signal amplitude. Available when **Superimpose Explicit Clock** is checked.
Explicit Clock Alignment	Sets the horizontal alignment of the superimposed clock waveform within the eye diagram. Available options are **Left** and **Center**. Left aligns the clock edge to the left boundary of the unit interval. Center aligns the clock edge to the center of the unit interval. Available when **Superimpose Explicit Clock** is checked.

Save panel fields and controls

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

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

Closing a plot view

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

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

PDF Eye plot configuration menu

This plot shows the underlying statistical model used to generate the BER contours.

PDF Eye Plot Configuration Menu

Use this menu to change settings of PDF Eye plot.

To open the PDF Eye plot configuration menu, double-tap anywhere in the PDF Eye 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 (PDF Eye plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Units	Sets the X-axis units to be unit intervals or seconds.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in BER units. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis scale	Sets the Y axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

BER Contour panel fields and Controls

Use this panel to control the BER Eye contours displayed on the Plot.

A checkbox corresponding to the six contours are provided. Target BER is the source measurement configuration Target BER value.

If checkbox is enabled, the contour to the specified Target BER will be displayed.

Save panel fields and controls

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

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

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

Closing a plot view

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

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

BER Eye plot configuration menu

This plot shows the probability of a hit vs the location in the eye.

BER Eye Plot Configuration Menu

Use this menu to change settings of BER Eye plot.

To open the BER Eye plot configuration menu, double-tap anywhere in the BER Eye 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 (BER Eye plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Units	Sets the X-axis units to be unit intervals or seconds.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in BER units. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.

Field or control	Description
Y-Axis scale	Sets the Y axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

BER Contour panel fields and Controls

Use this panel to control the BER Eye contours displayed on the Plot.

A checkbox corresponding to the six contours are provided. Target BER is the source measurement configuration Target BER value.

If checkbox is enabled, the contour to the specified Target BER will be displayed.

Save panel fields and controls

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

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

Closing a plot view

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

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

Correlated Eye plot configuration menu

This plot shows the data dependent eye with all uncorrelated effects removed.

Correlated Eye Plot Configuration Menu

Use this menu to change settings of Correlated Eye plot.

To open the Correlated Eye plot configuration menu, double-tap anywhere in the Correlated Eye 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 (Correlated Eye plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Units	Sets the X-axis units to be unit intervals or seconds.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in BER units. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis scale	Sets the Y axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.

BER Contour panel fields and Controls

Use this panel to control the Correlated Eye contours displayed on the Plot.

A checkbox corresponding to the six contours are provided. Target BER is the source measurement configuration Target BER value. If checkbox is enabled, the contour to the specified Target BER will be displayed.

Save panel fields and controls

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

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

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

Closing a plot view

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

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

BER Eye Contour plot configuration menu

This plot shows the Bit Error Rate contours at standard BER Levels like 1e-6, 1e-9, 1e-12, 1e-15, 1e-18 and Target BER on top of an accumulated eye diagram.

BER Eye Contour Plot Configuration Menu

Use this menu to change settings of BER Eye Contour plot.

To open the BER Eye Contour plot configuration menu, double-tap anywhere in the BER Eye Contour 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 (BER Eye Contour plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Units	Sets the X-axis units to be unit intervals or seconds.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in BER units. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.

Field or control	Description
Y-Axis scale	Sets the Y axis scale to Linear (default) or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Eye Rendering	Specifies the eye rendering mode as **Fast**, **Complete Vectors**, or **Complete Dots**. **Fast** renders the Unit Intervals (UIs) that define the eye boundaries, along with a specified number of surrounding UIs for additional visual context. **Complete Vectors** renders all valid UIs and interpolates the samples to produce smoother, higher-resolution rendering. **Complete Dots** renders all samples of the valid UIs without performing interpolation.
UIs	The user specified number of surrounding UIs for added visual context in Fast rendering. This option is visible only in **Fast rendering mode**.

Save panel fields and controls

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

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

Closing a plot view

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

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

Fast Frame Timestamp Trend plot configuration menu

Use this menu to change settings of a displayed Fast Frame Timestamp plot.

To open the Fast Frame Timestamp plot configuration menu, double-tap anywhere in a **Fast Frame Timestamp** Plot view.

Fast Frame Timestamp plot configuration menu fields and controls

Field or control	Description
AutoScale	Toggles AutoScale On or Off (Default is On). Turn AutoScale off to set the X and Y-axis range to view an area of interest. When AutoScale is Off (unchecked), the Plot view zoom mode enables and a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. You can also use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Selects which gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is Off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is Off.

Save panel fields and controls

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

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

Closing a plot view

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

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

Histogram plot configuration menu

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

To open the Histogram plot configuration menu, double-tap anywhere in a **Histogram** Plot view.

Settings panel (Histogram plot configuration menu) fields and controls

Field or control	Description
AutoScale	Toggles AutoScale on or off. Turn AutoScale off to set the X and Y-axis range to view an area of interest. When AutoScale is off, the Plot view zoom mode enables and a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Selects which gridlines to show in the plot.
X-Axis Number of Bins	Sets the resolution by the number of bins into which the X axis is divided.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales. These values also define the horizontal area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales. These values also define the vertical area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis Scale	Sets the Y axis scale to Linear or Log. A **Log** scale is useful when the component magnitudes cover a wide dynamic range, letting you show both lesser- and greater-magnitude components on the same display. A **Linear** scale is useful when the component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Plot Data	Sets the measurement data from the drop-down list to be displayed on plot. Available options are **Original** and **Emulated**.

Save panel fields and controls

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

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

Closing a plot view

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

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

Math FFT plot configuration menu (Math waveform)

Use the Math FFT plot menu to change settings of a displayed FFT plot, including source, FFT window type, plot type, and gating.

To open the FFT plot configuration menu, double-tap anywhere in the **Math FFT** Plot view.

Plot Settings panel (Math FFT plot configuration menu) fields and controls

Field or control	Description
Auto Scale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets which gridlines to show in the plot (Horizontal, Vertical, or Both).
X-Axis Scale	Sets the horizontal frequency scale to either Log or Linear. A **Log** scale is useful when the frequency component magnitudes cover a wide dynamic range, letting you show both lesser and greater- magnitude frequency components on the same display. A **Linear** scale is useful when the frequency component magnitudes are all close in value, allowing direct comparison of their magnitudes.
Y-Axis Scale	Sets the vertical amplitude scale to dBm or Vrms, when the FFT type is Magnitude. Sets the vertical amplitude scale to Linear, when the FFT type is Phase, Real or Imaginary. A **Log dB** scale is useful when the frequency component magnitudes cover a wide dynamic range, letting you show both lesser and greater magnitude frequency components on the same display. A **Linear** scale is useful when the frequency component magnitudes are all close in value, enabling direct comparison of their magnitudes.
X-Axis From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scale. These values also define the horizontal zoom area shown in the small Zoom window. Available when AutoScale is off.
Y-Axis From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scale. These values also define the vertical zoom area shown in the small Zoom window. Available when AutoScale is off.

FFT Settings panel (Math FFT plot configuration menu) fields and controls

Field or control	Description
FFT Type	**Magnitude** plots the magnitude values of the frequency components. **Phase** plots the phase of the signal as a function of frequency.
Window	Sets the FFT window type to use for the waveform plot. See table below.
Gating	Sets the region of the waveform to analyze for the FFT plot. **None** uses the entire waveform record to create the FFT plot. **Screen** uses the part of the waveform record displayed on the screen (such as in Zoom mode) to create the FFT plot. **Cursors** uses the waveform data between the cursors to create the FFT plot.
Vertical Units	Sets the vertical scale to Degrees, Radians, or Group Delay. **NOTE:** The Vertical Units set when FFT Type = Phase override the Y-Axis scale setting in the Plot Settings panel. Only available when FTT Type set to Phase.
Phase Wrap	When checked, the phase trace is unwrapped where the trace jumps more than the number of degrees set in the adjacent field. Only available when FTT Type set to Phase.
Squelch	When checked, the phase trace excludes points that have a voltage smaller than the specified voltage. The squelch voltage should be set to the expected noise voltage level. Minimum value is 100 mV, and increments in units of 100 mV. When unchecked, the phase trace includes all values. Only available when FTT Type set to Phase.

Save panel (Math FFT plot configuration menu) fields and controls

Field or control	Description
Save Plot Image	Opens the Save As (on page 366) configuration menu to specify the location and name at which to save an image file of the Plot view.
Save Plot Data	Opens the Save As (on page 367) configuration menu to specify the location and name at which to save the Plot view data to a CSV file.

FFT windows

Each FFT window is a trade-off between frequency resolution and magnitude accuracy. What you want to measure and your source signal characteristics help determine which window to use. Use the following guidelines to select the best window.

FFT window	Characteristics	Best for measuring
Hanning	Better frequency, poorer magnitude accuracy than Rectangular. Hanning has slightly poorer frequency resolution than Hamming.	Sine, periodic, and narrow-band random noise. Transients or bursts where the signal levels before and after the event are significantly different.
Rectangular	Best frequency, worst magnitude resolution. This is essentially the same as no window.	Transients or bursts where the signal levels before and after the event are nearly equal. Equal-amplitude sine waves with frequencies that are very close.
Hamming	Better frequency, poorer magnitude accuracy than Rectangular. Hamming has slightly better frequency resolution than Hanning.	Sine, periodic, and narrow-band random noise. Transients or bursts where the signal levels before and after the event are significantly different.
Blackman-Harris	Best magnitude, worst frequency resolution.	Predominantly single frequency signals to look for higher order harmonics.
Kaiser-Bessel	Less spectral leakage than the Hanning, Hamming, or Rectangular windows.	Predominantly single frequency signals to look for higher order harmonics.
Gaussian	The time-domain shape of an exponential Gaussian function transforms into a Gaussian exponential shape in the frequency domain.	Optimal localization in both the time and frequency domain.
Flattop2	Wider resolution bandwidth but lower side lobe attenuation. Also, it is unique because the time domain shape has negative values.	Useful for high accuracy magnitude measurements for signals that do not require very narrow bandwidth.
TekExp	The Tek Exponential window was invented at Tektronix. In the time domain, it is not asymmetrical bell shape as is the case with the other windows. Instead, it is exponential with a peak at the 20% position of the time domain gate. The frequency domain shape is triangular. More of the acquired data record length is used to capture the impulse response.	Use this window for impulse-response testing where the 20% position is the zero phase reference point.

You can also determine the best window empirically by first selecting the Rectangular window, and then selecting (in the following order) the Hamming, Hanning, and Blackman-Harris windows until the frequency components merge. Use the window just prior to where the frequencies emerge for the best compromise between resolution and amplitude accuracy.

Closing a plot view

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

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

Phase Noise plot configuration menu

Use this menu to change settings of a phase noise plot.

To open the Phase Noise plot configuration menu, double-tap anywhere in the **Phase Noise** 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 (Phase Noise plot configuration menu) fields and controls

Field or control	Description
Autoscale	Toggles auto scaling of the plot on or off. When AutoScale is off, a small Zoom window appears in the plot. Drag the blue zoom area box in the small Zoom window to view that area in the main Plot view. Use the Zoom and Pan front-panel knobs to change the zoom area and horizontal position.
Gridlines	Sets the gridlines to show in the plot. Available gridlines are Horizontal, Vertical, and Both.
X-Axis Scale From, To	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in unit intervals. These values also define the horizontal area displayed in the small Zoom window. Available when **AutoScale** is **Off**.
Y-Axis Scale From, To	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in unit intervals. These values also define the vertical area displayed in the small Zoom window. Available when **AutoScale** is **Off**.

Save panel fields and controls

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

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

Closing a plot view

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

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

Plot XY configuration menu

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

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

Settings panel (Plot XY configuration menu) fields and controls

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

- Use the vertical scale and position controls of the waveform connected to the X- and Y-axis sources to set horizontal scale and position of the XY waveform.
- XY format is particularly useful for studying phase relationships between two similar signals, creating Lissajous Patterns.
- XY format is a dot-only display, although it can have persistence. The Vector style selection has no effect when you select XY format.

Save panel fields and controls

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

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

Closing a plot view

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

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

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

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

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

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

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

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

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

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

To access the Save As configuration menu for plots:

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

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

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Opens the Browse Save As Location (on page 318) configuration menu to navigate to and select a location at which to save the file.
File name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Cancel	Cancels the file save action and closes the configuration menu.
OK	Saves the file to the specified location, closes the Save As menu, and displays a confirmation message.

Time Trend Plot configuration menu

Use the Time Trend configuration menu to set the trend plot waveform vertical scale and position, as well as add a label.

To open the Time Trend configuration menu, double-tap the **Trend** badge in the Settings bar. You can also double-tap the Time Trend Plot handle.

NOTE: Measurements that can use the time trend plot as a source will list the plot in the Source menu list.

Time Trend configuration menu fields and controls

Field or control	Description
Display	Toggles displaying the Trend plot **On** or **Off**.
Vertical Scale	Shows the vertical scale setting (when Auto Scale is on), or sets the vertical scale (when Auto Scale is off). Use the up and own arrow buttons to set the vertical scale value when Auto Scale is turned off. You can also use the multipurpose knob or screen keypad to change the value.
Auto Scale	Enables or disables the AutoScale mode. Auto Scale uses the signal data to dynamically set the vertical scale units.
Label	Adds a label to the Time Trend waveform (appears next to the Time Trend plot handle).

Field or control	Description
Position	Shows the vertical position setting (when Auto Scale is on), or sets the vertical position (when Auto Scale is off).
Set to 0	Sets the time trend plot to the center (zero) of the graticule (zero).
Save Plot Data	Saves the plot data to the specified location.

Deleting a trend plot

To delete a Time 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.

Transfer Function plot configuration menu

Use this menu to change settings of a displayed Transfer Function plot.

To open the Transfer Function plot menu, double-tap anywhere in the Transfer Function plot view.

To display a Transfer Function plot, see Display a Transfer Function plot (on page 98)

Settings panel (Transfer Function plot configuration menu) fields and controls

Field or control	Description
Numerator	Sets the measurement to use as the numerator (output) of the transfer function ratio. Select from the available measurements in the drop-down list.
Swap	Swaps the Numerator and Denominator selections, reversing the direction of the transfer function ratio.
Denominator	Sets the measurement to use as the denominator (input) of the transfer function ratio. Select from the available measurements in the drop-down list.
Smoothing	Toggles smoothing of the transfer function curve on or off. When On, the plot applies smoothing to reduce noise in the frequency-domain ratio, making trends in the response easier to interpret.
Gridlines	Selects which gridlines to display in the plot. Available options are **Horizontal**, **Vertical**, or **Both**. Select the grid style that meets your measurement needs.
AutoScale	Toggles AutoScale **On** or **Off**. Turn AutoScale off to manually set the X-axis and Y-axis ranges to view an area of interest. When AutoScale is off, a zoom window appears in the plot. Drag the zoom region to view the selected area in the main plot view. Use the Zoom and Pan controls to adjust the zoom area and position.
X-Axis (Frequency) **From, To**	Sets the beginning and end scale range to display in the plot for the X-Axis scales, in Hz. These values also define the horizontal area displayed in the small Zoom window. Available when **AutoScale** is **Off**.

Field or control	Description
Y-Axis (dB) **From, To**	Sets the beginning and end scale range to display in the plot for the Y-Axis scales, in dB. These values also define the vertical area displayed in the small Zoom window. Available when **AutoScale** is **Off**.

Save panel fields and controls

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

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

Closing a plot view

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

Deleting the numerator or denominator measurement badge will also close the plot.

XYZ plot configuration menu

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

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

Settings panel (Plot XYZ configuration menu) fields and controls

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

- XYZ format compares the voltage levels of the X and Y channel waveform records point-by-point, as in XY format. The displayed waveform intensity is modulated by the Z channel waveform amplitude.
- XYZ format is a dot-only display, although it can have persistence. The Vector style selection has no effect when you select XYZ format.

Save panel fields and controls

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

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

Closing a plot view

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

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

Reference waveform configuration menu

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

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

Reference waveform configuration menu fields and controls

Field or control	Description
Display	Turns **On** or **Off** displaying the waveform.
Vertical Scale	Set the vertical scale by using the assigned multipurpose knob, a virtual keypad, or tap the up or down arrows.
Label	Adds a label to the waveform. Tap and enter text using a keyboard, or double-tap to open the virtual keyboard. The label text is the same color as the waveform. Once you have entered the label, close the menu and double-tap the label text to open the Text Settings menu to change the font color, size, and other characteristics.
Units	Set the units label that you want to display on the vertical scale.
Position	Set the vertical position of the waveform using the assigned multipurpose knob or the virtual keypad.
Set to 0	Tap to set the vertical position to 0 (vertical center of the graticule).
Reference File	Shows the path and file name of the current Reference waveform. Double-tap the field to open the on-screen keyboard to enter or edit the path to open a different waveform file. Tap the down arrow icon to list the 20 most recently accessed reference waveform files.
Browse	Opens the Browse Waveform File dialog. Use this dialog to navigate to and select a waveform file to load. See Recall configuration menu (Ref waveform configuration menu) (on page 372).

Field or control	Description
Sample Rate, Record Length	Readout-only text that show the sample rate and record length values of the reference waveform.
Deskew	Changes the horizontal position of the reference waveform. Use this function to align the reference waveform to meet your measurement needs.
Set to 0	Sets the reference waveform deskew value to zero.
Selected Frame	Sets the frame to view of a recalled Fast Frame waveform file. This control is only available if the recalled waveform includes Fast Frame acquisitions. **NOTE:** This control has nothing to do with the selected frame of current acquired data. Adjusting this field does not change the selected frame in acquisition data. Conversely, adjusting the selected frame in the acquisition data does not change the selected frame of the reference waveform. Navigating through the frames of a reference waveform does not require the Fast Frame feature to be active. If more than one fast frame reference waveform is active, the Selected Frame in each waveform's configuration menu is independent of the others.

Recall configuration menu (Ref waveform configuration menu)

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

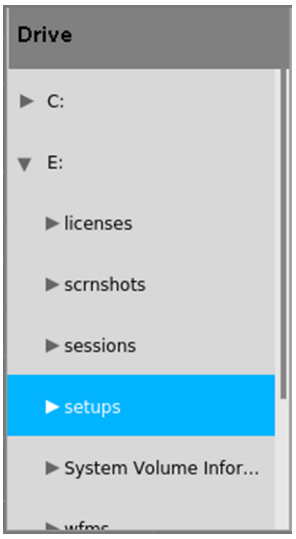
Prerequisite: A Ref badge must be present on the Settings bar. See Add a math, reference, or bus waveform (on page 82).

To open the Recall configuration menu:

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

Recall configuration menu (Ref configuration menu) fields and controls

Field or control	Description
Look in	Shows the current directory path and file name. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.

Field or control	Description
	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.
File Name	Lists the selected file name. Tap on the file path and use a keyboard to enter a new path. Or double-tap on the file name to open the on-screen keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recently accessed files, up to a maximum of 20.
Files of Type	Use to select the file format you want to open. Tap the field to show a list of all file extension types that the instrument can read for the selected file type. The files column only lists files of the specified type.
Cancel	Cancels any changes, closes the menu, and returns to the prior menu without loading a file.
OK	Closes the dialog, returns to the prior menu, and loads and displays the specified file.

Remote channel configuration menu

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

To open remote analog channel configuration menu, double-tap remote Channel badge.

The following text describes remote channel settings.

Vertical Settings panel, fields and controls (Remote Control On)

Field or control	Description
Display	Enables **(On)** or disables **(Off)** displaying the channel waveform in the Waveform View window.
Vertical Scale	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Invert	Toggles **On** or **Off**. Reverses the amplitude polarity of each sample point. When On, Invert adds a down arrow symbol to the channel badge.
Offset	Sets the channel signal vertical offset. When there is an offset, the remote channel handle displays it in the waveform view.
Set to 0	Sets the channel signal vertical offset to 0 units.
Position	Double-tap the field to set the vertical position using the virtual keypad.
Label	Double-tap the field to add a label to the channel display using the virtual keypad.

Vertical Settings panel, fields and controls (Remote Control Off)

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

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.

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.

Other search types

Edge Search configuration menu (on page 472)

Logic search configuration menu (on page 473)

Pulse Width Search configuration menu (on page 475)

Rise/Fall Time Search configuration menu (on page 476)

Runt Search configuration menu (on page 477)

Setup and Hold Search configuration menu (on page 478)

Timeout Search configuration menu (on page 480)

128b132b serial bus search configuration menu

Use the 128b132b search configuration menu to define conditions to search for and mark on 128b132b bus waveforms. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables the display of search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the 128b132b bus that you want to search.
Mark On	Select the type of information to search. Available mark on searches are Symbols, Sync Header, and Error.
Error Type	Sets the search on the Invalid sync.
Symbols	Uses the Symbols value configured in the Bus Configuration menu. The pattern varies according to the search. Available when **Mark On** is **Symbols**.
Pattern	Sets the pattern to search. Tap the Binary or Hex field and use the A and B knobs to select and change the values. You can also double-tap on the field to use the virtual keypad to enter values.

See also

Bus Search configuration menus (on page 375)

128b132b serial bus configuration menu (on page 205)

1-Wire serial bus search configuration menu

Use the 1-Wire Search configuration menu to define the conditions to search for and mark on a 1-Wire bus.

1-Wire search menu, fields, and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Search Type	Set to **Bus**.
Source	Select the 1-Wire bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates will show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
Reset	Searches and marks on the Reset event in the decode.
Presence	Select to notify the bus master that the device is on the bus and is ready to operate, and searches the same.
Command	Marks on the Memory Function Commands that are device-specific and are of 1 byte each (8 bits).
Data	Searches for user-defined data pattern in payload data.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Tap the field and use the A knob to change the value. Available only when **Mark On** is **Data**.
Read ROM	Allows to search for the family code and serial number of Read ROM packet. Available only when **Mode** is **Standard (15.4 Kbps)**.
Match ROM	Allows to search for the family code and serial number of Match ROM packet. Available only when **Mode** is **Standard (15.4 Kbps)**.
Skip ROM	Allows to search for the Skip ROM packet in the decode. Available only when **Mode** is **Standard (15.4 Kbps)**.
Search ROM	Allows to search the ROM numbers of the multiple devices. Available only when **Mode** is **Standard (15.4 Kbps)**.

Field or control	Description
ROM Code	Searches for the 64-bit ROM number in the search ROM packet. Available only when **Mode** is **Standard (15.4 Kbps)** and when **Mark On** is **Search ROM**.
Alarm Search	Allows to search the Alarm condition if present in the decode. Available only when **Mode** is **Standard (15.4 Kbps)**
Overdrive Match ROM	This command is same as the Match ROM command except that the device is only searched in overdrive mode. Available only when **Mode** is **Overdrive (125 Kbps)**.
Family Code	The least significant 8 bits of the address gives the family code of the device. Search can be performed on the same bits. Available only when **Mode** is **Standard (15.4 Kbps)** and when **Mark On is Read ROM, Match ROM**. Available only when **Mode** is **Overdrive (125 Kbps)** and when **Mark On** is **Overdrive Match ROM**.
Serial Number	The (next) least significant 48 bits give the serial number of the device and same can be searched. Available only when **Mode** is **Standard (15.4 Kbps)**. Available only when **Mark On** is**Read ROM, Match ROM**. Available only when **Mode** is **Overdrive (125 Kbps)** and when **Mark On** is **Overdrive Match ROM**.
Overdrive Skip ROM	This command is the same as the Skip ROM command except that only devices that can run in overdrive are searched. Available only when **Mode** is **Overdrive (125 Kbps)**.
CRC Error	Allows to search if any CRC error is present in the decode.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. 1-Wire is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. 1-Wire is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

1-Wire serial bus configuration menu (on page 206)

64B/66B serial bus search configuration menu

Use the 64B/66B search configuration menu to define conditions to search for and mark on 64B/66B bus waveforms. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables the display of search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See ActOnEventConfigMenu_3.xml#GUID-F418B032-8061-48B3-BF1C-953F1007C648 (on page 163) for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the 64B/66B bus that you want to search.
Mark On	Select the type of information to search. Available mark on searches are Symbols, Sync Header, and Error.
Error Type	Sets the search on the Invalid sync.
Symbols	Uses the Symbols value configured in the Bus Configuration menu. The pattern varies according to the search. Available when **Mark On** is **Symbols**.
Pattern	Sets the pattern to search. Tap the Binary or Hex field and use the A and B knobs to select and change the values. You can also double-tap on the field to use the virtual keypad to enter values.

See also

Bus Search configuration menus (on page 375)

64B/66B serial bus configuration menu (on page 207)

8b10b serial bus search configuration menu

Use the 8b10b Search configuration menu to define conditions to search for and mark on a 8b10b bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the 8b10b bus that you want to search.
Mark On	Select the type of information for which to search.
Format	Sets the number of data bytes for which to search (one to five bytes). Use the **A** knob to change the value. Available only when **Mark On** is **Symbols**.
Type	Sets the type to search (**Data** or **Control**). Available only when **Mark On** is **Symbols** and **Format** is **8-Bit**.

Field or control	Description
Disparity	Sets the disparity type as **Positive** or **Negative** or **Either**. Available only when **Mark On** is **Symbols** and **Format** is **8-Bit**, **10-Bit**, and **Symbol**. **NOTE:** Set the disparity value and then query the 10 bit data value.
Value	Sets the value for the format. ▪ When **Format** is **8-Bit**, you can set 8-Bit value in Binary or Hex format. ▪ When **Format** is **10-Bit**, you can set 10-Bit value in Binary or Hex format. ▪ When **Format** is **Symbol**, you can set values in decimal format. Available only when **Mark On** is **Symbols**.
10Bit RD+	Sets the RDpositive 10 bit value. It displays the corresponding 10 bit value for 8 bit or symbol. Set the Binary and Hex values Available when Format is 10 Bit
10Bit RD-	Sets the RDnegative 10 bit value. It displays the corresponding 10 bit value for 8 bit or symbol. Set the Binary and Hex values Available when Format is 10 Bit
Error Type	Sets the error type (**Symbol or Disparity**) for which to search and mark on the waveform. Available only when **Mark On** is **Errors**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. 8b10b is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. 8b10b is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

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.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.

Field or control	Description
Source	Select the **Audio** bus to search.
Mark On	Select the type of information for which to search.
Data	Sets the data pattern for which to search. Use in conjunction with the Mark When field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Data**.
Word	Sets the audio word channel for which to search. Available only when **Mark On** is **Data**.
Mark When	Sets the mark when condition for the specified data pattern. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

Audio serial bus configuration menu (on page 211)

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.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the **ARINC 429** bus to search.

Field or control	Description
Mark When	Sets the type of information for which to search.
Trigger When Label	Sets the label condition for which to search.
Label	Sets the label pattern for which to search. Tap the **Binary**, **Hex**, or **Octal** 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 **Mark When** is **Label** or **Label & Data**.
Label Low	Sets the low value of the label pattern range for which to search. Tap the **Binary**, **Hex**, or **Octal** 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 **Mark When** is **Label** and **Mark When Label** is **Inside Range** or **Outside Range**.
Label High	Sets the high value of the label pattern range for which to search. Tap the **Binary**, **Hex**, or **Octal** 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 **Mark When** is **Label** and **Mark When Label** is **Inside Range** or **Outside Range**.
Trigger When Data	Sets the data condition for which to search.
Data	Sets the data pattern for which to search. Tap the **Binary**, **Hex**, or **Octal** 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 **Mark When** is **Data** or **Label & Data**.
Data Low	Sets the low value of the data pattern range for which to search. Tap the **Binary**, **Hex**, or **Octal** 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 **Mark When** is **Data** and **Mark When Data** is **Inside Range** or **Outside Range**.
Data High	Sets the high value of the data pattern range for which to search. Tap the **Binary**, **Hex**, or **Octal** 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 **Mark When** is **Data** and **Mark When Data** is **Inside Range** or **Outside Range**.

Field or control	Description
SSM	Sets the Sign/Status Matrix (SSM) bit condition for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** is **Data** or **Label & Data**.
SDI	Sets the Source/Destination Identifier (SDI) bit condition for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark When** is **Data** or **Label & Data**.
Error Type	Sets the error condition for which to search. Available when **Mark When** is **Error**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

ARINC 429 serial bus menu (on page 209)

Auto Ethernet serial bus search configuration menu

Use the Auto Ethernet Search configuration menu to define conditions to search for and mark on an Auto Ethernet bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the **Auto Ethernet** bus that you want to search.

Field or control	Description
Mark On	Select the type of information for which to search. When the speed is **100 Base-T1**, the available options are **Start of Frame**, **Start of Frame Delimiter**, **MAC Address**, **Q-Tag Control Information**, **MAC Length/ Type**, **IP Header**, **TCP Header**, **Client Data**, **End of Packet**, **FCS (CRC) Error**. When the speed is **10 Base-T1S** and packet view is set to **On**, the available options are **MAC Address**, **Sync\Commit**, **Start of Stream Delimiter**, **Beacon**, **Ether Type**, **Payload**, **End of Stream Delimiter**, **End of Stream Delimiter OK**, and **Errors**. When the speed is **10 Base-T1S** and packet view is set to **Off**, the available options are **Sync\Commit**, **Start of Stream Delimiter**, **Beacon**, **Data**, **End of Stream Delimiter** and **End of Stream Delimiter OK**. When the speed is **1000 Base-T1,** mode is set to **Training**, and packet view is set to **On**, the available options are **TRAINING state**, **COUNTDOWN state**, and **Errors**. When the speed is **1000 Base-T1,** mode is set to **Training,** and packet view is set to **Off**, the available option is **Data**. When the speed is **1000 Base-T1,** mode is set to **Data,** packet view is set to **Off**, and **RS-FEC** is set to **Off,** the available options are **Header** and **Custom Pattern**. When the speed is **1000 Base-T1,** mode is set to **Data,** packet view is set to **Off**, and **RS-FEC** is set to **On,** the available options are **Header**, **FEC Parity**, **Custom Pattern**, and **Errors**. When the speed is **1000 Base-T1,** mode is set to **Data,** packet view is set to **On**, and **RS-FEC** is set to **On,** the available options are **MAC Address**, **MAC Length/Type**, **Ether Type**, and **Errors**. When the speed is **1000 Base-T1,** mode is set to **Data,** packet view is set to **On**, and **RS-FEC** is set to **Off,** the available options are **MAC Address**, **MAC Length/Type**, **Ether Type**, and **Errors**.
MAC Address Destination, MAC Address Source	Sets the MAC destination and/or source address pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is set to **Mac Address**.
Ether Type	Sets the ether type as **IPv4, IPv6,** or **ARP** to search. Available when **Mark On** is set to **Ether type**.
Header Type	Sets the header type for which to search. Available when **Mark On** is set to **Ether Type** . Available options are **ICMP**, **TCP**, **UDP**.
Payload Bytes	Sets the number of payload bytes. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is set to **Payload**.

Field or control	Description
Payload Value	Sets the payload value format used to display the bus information. Formats are Hex and Binary. Available when **Mark On** is set to **Payload**.
Q-Tag	Sets the Q-tag pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is set to **Q-Tag Control Information**.
MAC Length/Type	Enter the MAC length or type pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is set to **MAC Length/Type**.
IP Protocol	Sets the IP protocol pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is set to **IP Header**.
Source, Destination Address	Sets the source and/or destination IP address pattern for which to search. Tap a **Hex** or **Decimal** 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 **Mark On** is set to **IP Header**.
Source Port, Destination Port	Sets the source and/or destination TCP header port pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** 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 **Mark On** is set to **TCP Header**.
Sequence Number	Sets the TCP header sequence number pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** 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 **Mark On** is set to **TCP Header**.
Ack Number	Sets the TCP header ack number pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** 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 **Mark On** is set to **TCP Header**.

Field or control	Description
Data Bytes	Sets the number of client data bytes for which to search (one to sixteen bytes). Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is set to **Data** or **Client Data**.
Byte Offset	Sets the client data byte offset (**Don't Care** or the number of bytes). Tap the field and use the **A** knob to change the value. Available when **Mark On** is set to **Client Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary** and **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is set to **Data**.
Client Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex** and **ASCII** 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 **Mark On** is set to **Client Data**.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **MAC Length/Type** or **Client Data**. Available options are **=**, **≠**, **>**, **<**, **≥**, **≤**, **Inside Range**, and **Outside Range**.
Error Type	Sets the error type for which to search. Available when **Mark On** is set to **Errors**. Available options for 10Base-T1s are **CRC**, **ESDERR**, **ESDJAB**, **Checksum**, **Any**, and for 1000Base-T1 is **RS-FEC.**
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. Auto Ethernet is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. Auto Ethernet is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

Auto Ethernet serial bus configuration menu (on page 212)

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: See CAN serial bus search configuration menu (when using a .dbc symbol definition file) (on page 387) for CAN serial bus searches that use a loaded .dbc symbol definition file.

Field or control	Description
Display	Enables or disables displaying waveform search marks on this search.
Search Type	Set to **Bus**.
Source	Select the **CAN** bus that you want to search.
Mark On	Sets the type of information for which to search and mark on the bus waveform acquisition. **Start of Frame** sets a search mark at each occurrence of a CAN frame start. **Type of Frame** sets a search mark at each occurrence of the frame type as set in the Frame Type field. **Identifier** sets a search mark at each occurrence of a CAN identifier frame as set in the **Direction**, **Identifier Format**, and **Identifier** fields. Not available when the bus badge **Decode Format** is set to **Symbolic**. **Data** sets a search mark at the occurrence of the specified data fields (**Data Bytes**, **Data Offset**, **Mark When**, and **Data**). Not available when the bus badge **Decode Format** is set to **Symbolic**. **Id & Data** sets a search mark at each occurrence of the specified Id and data fields. Not available when the bus badge **Decode Format** is set to **Symbolic**. **End of Frame** sets a search mark at each occurrence of a CAN frame end. **Error** sets a search mark at each occurrence of the specified error condition (**Missing Ack**, **Bit Stuffing**, **FD Form Error**, or **Any Error**).
Frame Type	Sets the frame type for which to search and mark on the bus waveform (**Data Frame**, **Remote Frame**, **Error Frame**, **Overload Frame**). Available only when **Mark On** is **Type of Frame**.
Direction	Sets the Identifier's transfer direction (**Read**, **Write**, or **Either**) for which to search. Available only when **Mark On** is **Identifier**.
Identifier Format	Sets the CAN identifier format bit length for which to search (**Standard** 11-bit, or **Extended** 29-bit for CAN 2.0B). Available only when **Mark On** is **Identifier** or **ID & Data**.

Field or control	Description
Identifier	Sets the identifier pattern for which to search. The number of bits shown depends on the **Identifier Format** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Identifier** or **ID & Data**.
Data Bytes	Sets the number of data bytes for which to search (one to eight bytes). Use the **A** knob to change the value. Available only when **Mark On** is **Data** or **ID & Data**.
Data Offset	Sets the data offset value. Use the **A** knob to change the value. Available only when **Mark On** is **Data** or **ID & Data**.
Mark When	Sets the mark when condition (mark the waveform when the waveform is is, ≠, <, >, ≤, or ≥ the data conditions). Available only when **Mark On** is **Data** or **ID & Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Data** or **ID & Data**.
Error Type	Sets the error type for which to search and mark on the bus waveform. (**Missing Ack**, **Bit Stuffing**, or **Any Error**. Available only when **Mark On** is **Error**.
A, B knob controls	Use the **A** knob to select (highlight) the digit to change. Use the **B** knob to change the value of the digit.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

CAN serial bus configuration menu (on page 214)

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.

Field or control	Description
Display	Enables or disables displaying waveform search marks for this search.
Search Type	Set to **Bus**.
Source	Select the **CAN** bus that you want to search and mark.
Mark On	Sets the type of information for which to search and mark on the bus waveform acquisition. Available mark on options are Start of Frame, Types of Frame, FD Bits, Identifiers, Data, Id & Data, XL Bits, End of Frame, Error and Symbolic.
Symbolic	Sets the search for each occurrence of a specified message or signal condition as defined by an installed .dbc symbol definition file. This search setting is available when the **Decode Format** is set to **Symbolic** from CAN bus configuration menu.
Start of Frame	Sets the search for each occurrence of a CAN frame start.
End of Frame	Sets the search for each occurrence of a CAN frame end.
Type of Frame	Sets the frame type for which to search. This search setting is available when **Decode Format** is set to **Symbolic** from CAN bus configuration menu.
Configure	Opens the **CAN DBC Symbolic Configuration** menu, which sets the CAN bus search parameters based on the message and signal labeling derived from an installed .dbc symbol definition file.
Message, **Signal**, and **Value** fields	These fields display the actual search data values for the **Message**, **Signal**, and **Value** fields as set in the **CAN DBC Symbolic Configuration** menu. These fields are shown for reference and are not editable.
Data	Sets the data for which to search. Available when **CAN Standard** is in **CAN XL**, **CAN 2.0 + FD (ISO) + XL**, **CAN 2.0**, or **CAN FD (ISO)** except when **Decode Format** is set to **Symbolic** from CAN bus configuration menu.
ID & Data	Sets the ID & Data type for which to search. Available when **CAN Standard** is in **CAN 2.0 + FD (ISO) + XL**, **CAN 2.0**, or **CAN FD (ISO)** except when **Decode Format** is set to **Symbolic** from CAN bus configuration menu.
Identifier	Sets the identifier pattern on which to search. Available when **CAN Standard** is in **CAN 2.0 + FD (ISO) + XL**, **CAN 2.0**, or **CAN FD (ISO)** except when **Decode Format** is set to **Symbolic** from CAN bus configuration menu.
FD Bits	Sets a search mark at each occurrence of the specified **BRS Bit** (Bit Rate Switch) and **ESI Bit** (Error State Indicator) states. Available only when **CAN Standard** is in **CAN FD (ISO)** except when **Decode Format** is set to **Symbolic** from CAN bus configuration menu.

Field or control	Description
XL Bits	Sets the FD bits value on which to search. Available when **CAN Standard** is in **CAN XL** or **CAN 2.0 + FD (ISO) + XL** from CAN bus configuration menu.
BRS Bits	Sets the **BRS** (Bit Rate Switch) bit state for which to search and mark at each occurrence on the bus waveform. Available only when **Mark On** is **FD Bits**.
ESI Bits	Sets the **ESI** (Error State Indicator) bit state for which to search and mark at each occurrence on the bus waveform. Available only when **Mark On** is **FD Bits**.
Error Type	Sets the error type for which to search and mark on the bus waveform. (**Missing Ack**, **Bit Stuffing**, or **Any Error**. Available when **CAN Standard** is in **CAN 2.0 + FD (ISO) + XL**, **CAN 2.0**, or **CAN FD (ISO)** from CAN bus configuration menu and **Mark On** is set to **Error**.
A, B knob controls	Use the **A** knob to select (highlight) the digit to change. Use the **B** knob to change the value of the digit.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

CAN serial bus configuration menu (on page 214)

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.

Field or control	Description
Symbolic Type	Sets whether to search for **Message** CAN or search on both **Message and Signal** code.
Message	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.
Signal	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.

Field or control	Description
Signal Encoded	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.
Value	Sets a specific **Signal** 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 configuration menu (on page 214)

CPHY serial bus search configuration menu

Use the CPHY Search configuration menu to define the conditions to search for and mark on a CPHY bus.

CPHY search menu, fields, and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Search Type	Set to **Bus**.
Source	Select the CPHY bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates will show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
SoT	Searches the Start of Transmission (SoT) for each transmission. Available only when **SubType** is **CSI or DSI**.
EoT	Searches the End of Transmission (EoT) for each transmission. Available only when **SubType** is **CSI or DSI**.
Data	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 Mark, to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **SubType** is **CSI or DSI**.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Tap the field and use the A knob to change the value. Available only when **Mark On** is **Data**. Available only when **SubType** is **CSI or DSI**.

Field or control	Description
Mode	Sets the mode of operation to **HS** or **LP**. Available only when **Mark On** is **Data, Errors**. Available only when **SubType** is **CSI or DSI**.
Packet type	Sets the packet type for which to search. Available options are **Short** or **Long**. Default is **Short**. Available only when **Mark On** is **Packets**. Available only when **SubType** is **CSI or DSI**.
Packet Name	Sets the packet name for which to search. Available options are **Frame Start, Frame End, Line Start, Line End, Generic Short**. Available only when **Packet Type**is **Short** and **SubType** is **CSI**. Available options are **V Sync Start, V Sync End, H Sync Start, H Sync End, End Of Transmission, Color Mode Off, Color Mode On, Shutdown Peripheral, Turn On Peripheral, Generic Short Write, Generic Read, Compression, Scrambling, Execute Queue, DCS Short Write, DCS Read, DCS Long Write**. Available only when **Packet Type**is **Short** and **SubType is DSI**. Available options are **Null, Blanking, Embedded, User 8-Bit, RGB565, RGB555, RGB444, RGB666, RGB888, RAW6, RAW7, RAW8, RAW10, RAW12, RAW14, RAW16, RAW20, YUV420 8-bit (Legacy), YUV 8-Bit, YUV422 10-bit, YUV420 10-bit**. Available only when **Packet Type is Short** and **SubType is CSI**. Available options are **DCS Long Write, Null, Blanking, Generic Long Write, Picture Parameter Set, Compressed Pixel Stream, Packed Pixel 30-bit RGB 10-10-10, Packed Pixel 36-bit RGB 12-12-12, Packed Pixel 16-bit RGB 5-6-5, Packed Pixel 18-bit RGB 6-6-6, Loosely Packed Pixel 18-bit RGB 6-6-6, Packed Pixel 24-bit RGB 8-8-8, Packed Pixel 24-bit YCbCr 4:2:2, Loosely Packed 24-Bit YCbCr 4:2:2, Packed Pixel 16-bit YCbCr 4:2:2, Packed Pixel 12-bit YCbCr 4:2:0**. Available only when **Packet Type** is **Short** and **SubType is DSI**. Available only when **Mark On is Packets**.
Escape Mode	Search for the escape mode. Available only when **Packet Type**is **Short** and **SubType** is **CSI or DSI**.
Command	Available options are **Low-Power Data Transmission, Ultra-Power State, Reset Trigger, Any**. Available only when **Mark On** is **Escape Mode**. Available only when **SubType** is **CSI or DSI**.

Field or control	Description
Stop	Search for escape mode exit. Available only when **SubType** is **CSI or DSI**.
Error Type	Sets the error type for which to search. Available options are **Any, ECC, CRC**. Default is **Any**. Available only when **Mark On** is **Errors**. Available only when **SubType** is **CSI or DSI**.
Optional parameter	Search for an optional parameter. Available only when **SubType** is **DSI**.
Scrambling	Search the scrambling mode command.
Compression	Search for the compression mode command. Available only when **SubType** is **DSI**.
BusTurnAround	Searches the bus turn around.
Word	Sets the word packet pattern for which to search. The number of bits shown depends on the Data words setting. Use in conjunction with the Mark, to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **SubType** is **Word**.
Data Words	Sets the number of data word for which to search (one to five bytes). Tap the field and use the A knob to change the value. Available only when **Mark On** is **Word**.
Symbol	Sets the symbol pattern for which to search. The number of bits shown depends on the No. of symbols setting. Use in conjunction with the Mark, to specify the exact search condition. Tap the **Binary** or **Oct** 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 only when **SubType** is **Symbol**.
No. of Symbols	Sets the number of symbols for which to search (one to five bytes). Tap the field and use the A knob to change the value. Available only when **Mark On** is **Symbol**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. CPHY is currently not supported as a bus trigger source.

Field or control	Description
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. CPHY is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

CPHY serial bus configuration menu (on page 217)

CXPI serial bus search configuration menu

Use the CXPI Search configuration menu to define the conditions to search for and mark on a CXPI bus.

CXPI search search menu, fields, and controls

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the CXPI bus that you want to search.
Mark On	Select the type of information for which to search.
Start	Select to search the start event of the packet decode.
Frames	Select to search on the different frames.
Frame Type	Sets the frame type. Available options are **Normal, Sleep, Long, Polling Normal, Polling long**. Default is **Normal**. Available when **Mark On** is **Frames**
Frame ID	Sets the frame id value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Frame ID**.
PTYPE	Select to search for PTYPE (Protected Type) field.
DLC	Sets the DLC value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **DLC**.

Field or control	Description
Ext DLC	Sets the Ext DLC value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Ext DLC**.
Network Management	Select to search for wakeup and sleep function of the slave nodes used to save the electrical power consumption.
Wakeup.Ind	Sets the wakeup Ind type. Switches between **On, Off,** or, **Either**. Default is **On**.
Sleep.Ind	Sets the Sleep Ind type. Switches between **On, Off,** or, **Either**. Default is **On**.
Counter	Sets the counter value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Counter**.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Tap the field and use the A knob to change the value. Available only when **Mark On** is **Data**.
Data	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 Mark, to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Data**.
Error Type	Sets the error type for which to search. Available options are **CRC, Parity, Frame, IBS, Any.** Default is **Parity**. Available only when **Mark On** is **Errors**.

See also

Bus Search configuration menus (on page 375)

CXPI serial bus configuration menu (on page 218)

Configure custom pattern editor

Use the symbol keypad to configure the custom pattern comprising of Data/K-Code values. To open the keypad, double-tap inside the character/hex fields in the custom pattern editor.

K-Code pattern editor

Click the toggle button to **K-Code** from the switch settings menu.

K-Code keypad fields and controls

Keys	Description
COM (K28.5)	Sets the K-Code to K28.5
EDB (K28.3)	Sets the K-Code to K28.3
END (K29.7)	Sets the K-Code to K29.7
EPF (K23.7)	Sets the K-Code to K23.7
SDP (K28.2)	Sets the K-Code to K28.2
SHP (K27.7)	Sets the K-Code to K27.7
SKP (K28.1)	Sets the K-Code to K28.1
SLC (K30.7)	Sets the K-Code to K30.7
SUB (K28.4)	Sets the K-Code to K28.4
K28.0	Sets the K-Code to K28.0
K28.6	Sets the K-Code to K28.6
K28.7	Sets the K-Code to K28.7
Clear	Clears the K-Code in the Custom Pattern Editor.
Cancel	Closes the keypad without updating the Custom Pattern Editor.
Enter	Updates the selected K-Code to the Custom Pattern Editor's character field and closes the keypad.

Data pattern editor

Tap to toggle button to **Data** from the switch settings menu.

Data keypad fields and controls

Keys	Description
Entry field	Enter the text using virtual keypad. It shows the Hex value (XX) 0-9, A-F, and X.
0-9 and A-F and X	Sets the value to the selected text in the entry field.
Clear	Clears the entry field value and sets it to XX.
Left Arrow	Moves the selection in the entry field to the left.
Right Arrow	Moves the selection in the entry field to the right.
Cancel	Closes the keypad without updating the Editor.

Keys	Description
Enter	Updates the value in the Entry field to the Custom Pattern Editor's hex field and closes the keypad.

NOTE: A minimum of one pattern and maximum of 16 patterns can be added.

Configure pattern editor

Use the pattern editor keypad to configure the pattern for the data or custom. To open the pattern editor, double-tap inside a Data/Custom field that requires alphanumeric values.

Data Pattern Editor

Keys	Description
Entry field	Enter the Data and hex values (D-XX). D is the data and XX is the hex value which can be 0-9 and A-F.
0-9 and A-F	Sets the Hex value.
CLEAR	Clears all values from the input entry field.
Bksp	Deletes characters to the left of the insert text marker position.
Cancel	Cancels all the values from the input entry field.
Enter	Cancels all the values from the input entry field.

Custom Pattern Editor

Keys	Description
NULL	Enters N in the input box.
FCT	Enters F in the input box.
EOP	Enters EOP in the input box.
EEP	Enters EE in the input box.
T-XX	Sets the timecode and the hex values (T-XX). T is the Timecode and XX is the hex value which can be 0-9 and A-F.
D-XX	Sets the data and the hex values (T-XX). T is the data and XX is the hex value which can be 0-9 and A-F.
0-9 and A-F	Sets the Hex value.
CLEAR	Clears all values from the input entry field.
Bksp	Cancels all the values from the input entry field.
Cancel	Cancels all the values from the input entry field.

Keys	Description
Enter	Cancels all the values from the input entry field.

NOTE: A minimum of two patterns and maximum of eight patterns can be added. Each pattern should be separated by a space.

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
CS	Sets the chip select source, level, and the threshold value.
RAS	Sets the row access strobe source, level, and the threshold value.
CAS	Sets the column access strobe source, level, and the threshold value.
WE	Sets the write enable source, level, and the threshold value.

DDR Read Search configuration menu

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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **DDR Read**.
Source	Lists the source channel or waveform to use to search.
Standard	Sets the DDR standard (**DDR3** or **LPDDR3**).

Field or control	Description
Data Rate	Sets the data rate options for the selected standard. Data rate values for the **DDR3** standard are 800, 1066, 1333, 1600, 1866, 2133, Auto, and Custom. Default is 800. Data rate values for the **LPDDR** standard are 333, 800, 1066, 1200, 1333, 1466, 1600, 1866, 2133, Auto, and Custom. Default is 333.
Custom Data Rate	Sets the custom date rate. Default is 800 MT/s. Available only when **Data Rate** is **Custom**.
Burst Detection Method	Sets the burst detection method to use for the measurement. ▪ DQ/DQS Phase Alignment ▪ Chip Select, Latency + DQ/DQS Phase Alignment ▪ Logic State + Burst latency
Postamble Length (tCK)	Sets the postamble length value. Default is 500 mtCK.
Preamble Type	Sets the preamble type (**Static** or **Dynamic**). Default is **Static**.
DQ/DQS Levels	Sets to **Auto** or **Manual**. Default is **Auto**.
Configure	Sets the DQ/DQS level values. See DQ/DQS Levels Configure menu (on page 405). Available only when **DQ/DQS Levels** is **Manual**.
Source 3 (CS)	Sets the CS (Chip Select) signal source. Default is Ch 4. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.
CAS Min	Sets the CAS (Column Access Strobe) minimum cycles value. Default value is 2.0 cycles. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.
CAS Max	Sets the CAS maximum cycles value. Default value is 3.0 cycles. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**. **NOTE:** 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.
CS Mode	Sets the CS (Chip Select) to Auto or Manual mode. Default is Auto. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.

Field or control	Description
CS Level	Sets the CS (Chip Select) level value. Default is 500 mV. Available only when **CS Mode** is **Manual**.
CS Active	Sets the CS signal active true state to Low or High. Default is Low. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.
Logic Pattern	Sets the Logic pattern values. See DDR Input Configure menu (on page 397). Available only when **Burst Detection Method** is **Logic State + Burst latency** .
Burst Latency	Sets the burst latency value. Default is 2.5 Cycles. Available only when **Burst Detection Method** is **Logic State + Burst latency** .
Tolerance	Sets the tolerance value. Default is 1 Cycle.
Burst Length	Sets the burst length. Default is 8 UI.

DDR Read & Write Search configuration menu

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 & Write Search configuration menu fields and controls

Field or control	Description
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **DDR Read**.
Source	Lists the source channel or waveform to use to search.
Standard	Sets the DDR standard (**DDR3** or **LPDDR3**).

Field or control	Description
Data Rate	Sets the data rate options for the selected standard. Data rate values for the **DDR3** standard are 800, 1066, 1333, 1600, 1866, 2133, Auto, and Custom. Default is 800. Data rate values for the **LPDDR** standard are 333, 800, 1066, 1200, 1333, 1466, 1600, 1866, 2133, Auto, and Custom. Default is 333.
Custom Data Rate	Sets the custom date rate. Default is 800 MT/s. Available only when **Data Rate** is **Custom**.
Burst Detection Method	Sets the burst detection method to use for the measurement. ▪ DQ/DQS Phase Alignment ▪ Chip Select, Latency + DQ/DQS Phase Alignment ▪ Logic State + Burst latency
Postamble Length (tCK)	Sets the postamble length value. Default is 500 mtCK.
Preamble Type	Sets the preamble type (**Static** or **Dynamic**). Default is **Static**.
DQ/DQS Levels	Sets to **Auto** or **Manual**. Default is **Auto**.
Configure	Sets the DQ/DQS level values. See DQ/DQS Levels Configure menu (on page 405). Only available when **DQ/DQS Levels** is **Manual**.
Source 3 (CS)	Sets the CS (Chip Select) signal source. Default is Ch 4. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.
CAS Min	Sets the CAS (Column Access Strobe) minimum cycles value. Default value is 2.0 cycles. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.
CAS Max	Sets the CAS maximum cycles value. Default value is 3.0 cycles. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**. **NOTE:** 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.
CS Mode	Sets the CS (Chip Select) to Auto or Manual mode. Default is Auto. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.

Field or control	Description
CS Level	Sets the CS (Chip Select) level value. Default is 500 mV. Available only when **CS Mode** is **Manual**.
CS Active	Sets the CS signal active true state to Low or High. Default is Low. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.
Logic Pattern	Sets the Logic pattern values. See DDR Input Configure menu (on page 397). Available only when **Burst Detection Method** is **Logic State + Burst latency**.
Burst Latency	Sets the burst latency value. Default is 2.5 Cycles. Available only when **Burst Detection Method** is **Logic State + Burst latency** .
Tolerance	Sets the tolerance value. Default is 1 Cycle.
Burst Length	Sets the burst length. Default is 8 UI.

DDR Write Search configuration menu

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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **DDR Read**.
Source	Lists the source channel or waveform to use to search.
Standard	Sets the DDR standard (**DDR3** or **LPDDR3**).

Field or control	Description
Data Rate	Sets the data rate options for the selected standard. Data rate values for the **DDR3** standard are 800, 1066, 1333, 1600, 1866, 2133, Auto, and Custom. Default is 800. Data rate values for the **LPDDR** standard are 333, 800, 1066, 1200, 1333, 1466, 1600, 1866, 2133, Auto, and Custom. Default is 333.
Custom Data Rate	Sets the custom date rate. Default is 800 MT/s. Available only when **Data Rate** is **Custom**.
Burst Detection Method	Sets the burst detection method to use for the measurement. ▪ DQ/DQS Phase Alignment ▪ Chip Select, Latency + DQ/DQS Phase Alignment ▪ Logic State + Burst latency
Postamble Length (tCK)	Sets the postamble length value. Default is 500 mtCK.
Preamble Type	Sets the preamble type (**Static** or **Dynamic**). Default is **Static**.
DQ/DQS Levels	Sets to **Auto** or **Manual**. Default is **Auto**.
Configure	Sets the DQ/DQS level values. See DQ/DQS Levels Configure menu (on page 405). Only available when **DQ/DQS Levels** is **Manual**.
Source 3 (CS)	Sets the CS (Chip Select) signal source. Default is Ch 4. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.
CAS Min	Sets the CAS (Column Access Strobe) minimum cycles value. Default value is 2.0 cycles. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.
CAS Max	Sets the CAS maximum cycles value. Default value is 3.0 cycles. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**. **NOTE:** 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.
CS Mode	Sets the CS (Chip Select) to Auto or Manual mode. Default is Auto. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.

Field or control	Description
CS Level	Sets the CS (Chip Select) level value. Default is 500 mV. Available only when **CS Mode** is **Manual**.
CS Active	Sets the CS signal active true state to Low or High. Default is Low. Available only when **Burst Detection Method** is **Chip Select, Latency + DQ/DQS Phase Alignment**.
Logic Pattern	Sets the Logic pattern values. See DDR Input Configure menu (on page 397). Available only when **Burst Detection Method** is **Logic State + Burst latency** .
Burst Latency	Sets the burst latency value. Default is 2.5 Cycles. Available only when **Burst Detection Method** is **Logic State + Burst latency** .
Tolerance	Sets the tolerance value. Default is 1 Cycle.
Burst Length	Sets the burst length. Default is 8 UI.

DPHY serial bus search configuration menu

Use the DPHY Search configuration menu to define the conditions to search for and mark on an DPHY bus.

DPHY search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Search Type	Set to **Bus**.
Source	Select the DPHY bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates will show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
SoT	Select the Start of Transmission (SoT) for each transmission.
EoT	Select the End of Transmission (EoT) for each transmission.
Mode	Set the mode of operation to HS or LP. Available only when **Mark On** is **Data**, **Scrambling**, **Compression**, **Packets**, and **Errors**.

Field or control	Description
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Tap the field and use the A knob to change the value. Available only when **Mark On** is **Data**.
Data	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 Mark When field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Data**.
Scrambling	Search the scrambling mode command.
Compression	Search for the compression mode command.
Packet Type	Sets the packet type for which to search. Available options are **Short** or **Long**. Default is Short. Available only when **Mark On** is **Packets**.
Packet Name	Sets the packet name for which to search. Available only when **Mark On** is **Packets**.
BusTurnAround	Searches the bus turn around.
Escape	Search for the escape entry mode.
Stop	Search for escape mode exit.
Error Type	Sets the error type for which to search. Available when **Mark On** is **Error**. Available options are **Any**, **ECC**, or **CRC**. Default is Any.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. DPHY is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. DPHY is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

DPHY serial bus configuration menu (on page 219)

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
High	Sets the high value for Strobe and Data. Default value is 0.
Mid	Sets the mid value for Strobe and Data. Default value is 0.
Low	Sets the low value for Strobe and Data. Default value is 0.
Hysteresis	Sets the hysteresis. Default value is 10%.
Margin	Sets the margin. Default value is 20%.

eSPI serial bus search configuration menu

Use the eSPI Search configuration menu to define the conditions to search for and mark on a eSPI bus.

eSPI search menu, fields, and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Search Type	Set to **Bus**.
Source	Select the eSPI bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates will show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
Start	Enables to search the start event of the packet decode.
Channel Independent	Enables search on Channel Independent command or response packets.
Peripheral Channel	Enables search on Peripheral Channel command or response packets.
OOB Channel	Enables search on OOB Channel command or response packets.
Virtual Wire Channel	Enables search on Virtual Wire Channel command or response packets.
Flash Access Channel	Enables search on Flash Access Channel command or response packets.
Wait	Enables to search on the wait state that appears after the TAR window.

Field or control	Description
End	Enables to search on the end events when the packet decode ends.
Phase	Select the type of phase between command and response packets. When **Mark On** is **Channel Independent** or **OOB Channel** or **Data** or **Errors**. Available options are **Command** and **Response**. Default is **Command**. When **Mark On** is **Peripheral Channel** or **Virtual Wire Channel** or **Flash Access Channel**. Available options are **Command**, **Response With Header** and **Response Without Header**. Default is **Command**.
Command	Enables search for the command packets.
Response	Enables search for the response packets.
Response With Header	Enables to search on the RSP opcode that consists of a Response Code, Reserved, and Response Modifier. The size is 8 bits. It includes the optional HDR/optional data appended after it.
Response Without Header	Enables to search on the RSP opcode that consists of a Response Code, Reserved, and Response Modifier. The size is 8 bits.
Command Opcode	Enables search on the command opcode of different channels specified under the mark on. The size is 8 bits. When **Mark On** is **Channel Independent** and **Phase** is **Command**, available options are **Get Status**, **Get Configuration**, **Set Configuration**. Default is **Get Status**. When **Mark On** is **Peripheral Channel** and **Phase** is **Command**, available options are **GET_NP**, **GET_PC**, **PUT_NP**, **PUT_PC**, **PUT_IORD_SHORT**, **PUT_IOWR_SHORT**, **PUT_MEMRD32_SHORT**, **PUT_MEMWR32_SHORT**. Default is **GET_NP**. When **Mark On** is **OOB Channel** and **Phase** is **Command**, available options are **PUT_OOB**, **GET_OOB**. Default is **PUT_OOB**. When **Mark On** is **Virtual Wire Channel** and **Phase** is **Command**, available options are **PUT_VWIRE**, **GET_VWIRE**. Default is **PUT_VWIRE**. When **Mark On** is **Flash Access Channel** and **Phase** is **Command**, available options are **PUT_FLASH_C**, **PUT_FLASH_NP**, **GET_FLASH_C**, **GET_FLASH_NP**. Default is **PUT_FLASH_C**.

Field or control	Description
Cycle Type	Enables search under command and response with header based on different cycle types for different channels. The size is 8 bits. When **Mark On** is **Peripheral Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_NP**, available options are **Memory Read 32**, **Memory Read 64**. Default is **Memory Read 32**. When **Mark On** is **Peripheral Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_PC**, available options are **Memory Write 32**, **Memory Write 64**, **Message**, **Message With Data**, **Successful Completion With Data**, **Unsuccessful Completion With Data**. Default is **Memory Write 32**. When **Mark On** is **Peripheral Channel**, **Phase** is **Response With Header**, available options are **Memory Read/Write 32**, **Memory Read/Write 64**, **Message**, **Message With Data**, **Successful Completion Without Data**, **Successful Completion With Data**, **Unsuccessful Completion With Data**, **Message With LTR**. Default is **Memory Read /Write 32**. When **Mark On** is **OOB Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_OOB**, available options are **OOB (Tunneled SMBus) Message**, **MTCP**. Default is **OOB (Tunneled SMBus) Message**. When **Mark On** is **Flash Access Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_FLASH_C**, available options are **Successful Completion Without Data**, **Successful Completion With Data**, **Unsuccessful Completion With Data**. Default is **Unsuccessful Completion Without Data**. When **Mark On** is **Flash Access Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_FLASH_NP**, available options are **RPMC OP1 Successful Completion With Data**, **RPMC OP1 Successful Completion Without Data**, **RPMC OP2 Successful Completion With Data**, **RPMC OP2 Successful Completion Without Data**, **RPMC OP1 OP2 Unsuccessful Completion Without Data**. Default is **RPMC OP1 Successful Completion With Data.** When **Mark On** is **Flash Access Channel**, **Phase** is **Response With Header**, available options are **Flash Read**, **Flash Write**, **Flash Erase**. Default is **Flash Read**.

Field or control	Description
Address	Enables search on the address field for different channels based on different commands and response with header classified based on cycle types. Address field can vary anywhere between 8bits/16bits/32bits or 64bits. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Channel Independent**, **Phase** is **Command**, **Command Opcode** is **Get Configuration**, **Set Configuration**. Available only when **Mark On** is **Peripheral Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_NP**, **Cycle Type** is **Memory Read 32**, **Memory Read 64**. Available only when **Mark On** is **Peripheral Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_PC**, **Cycle Type** is **Memory Write 32**, **Memory Write 64**. Available only when **Mark On** is **Peripheral Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_IORT_SHORT**, **PUT_IOWR_SHORT**, **PUT_MEMRD32_SHORT**, **PUTMEMWR32_SHORT**. Available only when **Mark On** is **Peripheral Channel**, **Phase** is **Response With Header**, **Cycle Type** is **Memory Read/Write 32**, **Memory Read/Write 64**. Available only when **Mark On** is **OOB Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_OOB**, **Cycle Type** is **OOB (Tunneled SMBus) Message**, **MTCP**. Available only when **Mark On** is **Flash Access Channel**, **Phase** is **Response With Header**, **Cycle Type** is **Flash Read**, **Flash Write**, **Flash Erase**.

Field or control	Description
Tag	Enables search on the tag field for different channels based on different commands and response with header classified based on cycle types. The size is 4 bits. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Peripheral Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_NP**, **Cycle Type** is **Memory Read 32**, **Memory Read 64** . Available only when **Mark On** is **Peripheral Channel**,**Phase** is **Command**, **Command Opcode** is **PUT_PC**, **Cycle Type** is **Memory Write 32**, **Memory Write 64**, **Message**, **Message With Data**, **Successful Completion With Data**, **Unsuccessful Completion With Data**. Available only when **Mark On** is **Peripheral Channel**, **Phase** is **Response With Header**, **Cycle Type** is **Memory Read/Write 32**, **Memory Read/Write 64**, **Message**, **Message With Data**, **Successful Completion Without Data**, **Successful Completion With Data**, **Unsuccessful Completion With Data**, **Message With LTR**. Available only when **Mark On** is **OOB Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_OOB**, **Cycle Type** is **OOB (Tunneled SMBus)**, **MCTP**. Available only when **Mark On** is **OOB Channel**, **Phase** is **Response**. Available only when **Mark On** is **Flash Access Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_FLASH_C**, **Cycle Type** is **Successful Completion Without Data**, **Successful Completion With Data**, **Unsuccessful Completion With Data**. When **Mark On** is **Flash Access Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_FLASH_NP**, available options are **RPMC OP1 Successful Completion With Data**, **RPMC OP1 Successful Completion Without Data**, **RPMC OP2 Successful Completion With Data**, **RPMC OP2 Successful Completion Without Data**, **RPMC OP1 OP2 Unsuccessful Completion Without Data**. Available only when **Mark On** is **Flash Access Channel**, **Phase** is **Response With Header**, **Cycle Type** is **Flash Read**, **Flash Write**, **Flash Erase**.

Field or control	Description
Length	Enables search on the length field for different channels based on different commands and response with header classified based on cycle types. The size is 12 bits. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_NP**, **Cycle Type** is **Memory Read 32**, **Memory Read 64**. Available only when **Mark On** is **Peripheral Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_PC**, **Cycle Type** is **Memory Write 32**, **Memory Write 64**, **Message**, **Message With Data**, **Successful Completion With Data**, **Unsuccessful Completion With Data**. Available only when **Mark On** is **Peripheral Channel**, **Phase** is **Response With Header**, **Cycle Type** is **Memory Read/Write 32**, **Memory Read/Write 64**, **Message**, **Message With Data**, **Successful Completion Without Data**, **Successful Completion With Data**, **Unsuccessful Completion With Data**. Available only when **Mark On** is **OOB Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_OOB**, **Cycle Type** is **OOB (Tunneled SMBus)**, **MCTP**. Available onlywhen **Mark On** is **OOB Channel**, **Phase** is **Response**. Available only when **Mark On** is **Flash Access Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_FLASH_C**, **Cycle Type** is **Successful Completion Without Data**, **Successful Completion With Data**, **Unsuccessful Completion With Data**. When **Mark On** is **Flash Access Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_FLASH_NP**, available options are **RPMC OP1 Successful Completion With Data**, **RPMC OP1 Successful Completion Without Data**, **RPMC OP2 Successful Completion With Data**, **RPMC OP2 Successful Completion Without Data**, **RPMC OP1 OP2 Unsuccessful Completion Without Data**. Available only when **Mark On** is **Flash Access Channel**, **Phase** is **Response With Header**, **Cycle Type** is **Flash Read**, **Flash Write**, **Flash Erase**.
SMBus Slave Address	Enables search on SMBus Slave address under the OOB channel. The size is 8 bits. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **OOB Channel**, **Phase** is **Response**.
Virtual Wire Count	Enables search on Virtual Wire Count for command and response with header under the virtual wire channel. The size is 8 bits. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Virtual Wire Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_VWIRE**. Available only when **Mark On** is **Virtual Wire Channel**, **Phase** is **Response With Header**.

Field or control	Description
Virtual Wire Index	Enables search on Virtual Wire index for command and response with header under the virtual wire channel. The size is 8 bits. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Virtual Wire Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_VWIRE**. Available only when **Mark On** is **Virtual Wire Channel**, **Phase** is **Response With Header**.
Virtual Wire Data	Enables search on Virtual Wire Data for command and response with header under the virtual wire channel. The size is 8 bits. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Virtual Wire Channel**, **Phase** is **Command**, **Command Opcode** is **PUT_VWIRE**. Available only when **Mark On** is **Virtual Wire Channel**, **Phase** is **Response With Header**.
Data Bytes	Sets the number of data bytes for which to search. 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 a value. Available only when **Mark On** is **Data**.
Data	Sets the data value for which to search. Searches based on command or response phase. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Data**.
Status	Enables search on the status field of the response packets. The size is 16 bits. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Status**.
Error Type	Sets the error type for which to search. When **Phase** is **Command**, available options are **CRC**, **Command Opcode**, **Cycle Type**. When **Phase** is **Response**, available options are **CRC**, **Defer**, **Fatal**, **Non-Fatal**, **No Response**. Available only when **Mark On** is **Errors**.

See also

Bus Search configuration menus (on page 375)

eSPI serial bus configuration menu (on page 220)

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.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the Ethernet bus that you want to search.
Mark On	Select the type of information for which to search.
MAC Address Destination, MAC Address Source	Sets the MAC destination and/or source address pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** is **Mac Addresses**.
Q-Tag	Sets the Q-tag pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** is **Q-Tag Control Information**.
MAC Length/Type	Enter the MAC length or type pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when M**ark On** = **MAC Length/Type**.
IP Protocol	Sets the IP protocol pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** is **IP Header**.
Source, Destination Address	Sets the source and/or destination IP address pattern for which to search. Tap a **Hex** or **Decimal** 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. Only available when **Mark On** is **IP Header**.

Field or control	Description
Source Port, Destination Port	Sets the source and/or destination TCP header port pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** 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. Only available when **Mark On** is **TCP Header**.
Sequence Number	Sets the TCP header sequence number pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** 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. Only available when **Mark On** is **TCP Header**.
Ack Number	Sets the TCP header ack number pattern for which to search. Tap a **Binary**, **Hex** or **Decimal** 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. Only available when **Mark On** is **TCP Header**.
Data Bytes	Sets the number of client data bytes for which to search (one to sixteen bytes). Use the **A** knob to change the value. Only available when **Mark On** is **Client Data**.
Byte Offset	Sets the client data byte offset (**Don't Care** or the number of bytes). Tap the field and use the **A** knob to change the value. Only available when **Mark On** is **Client Data**.
Client Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex** or **ASCII** 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. Only available when **Mark On** is **Client Data.**
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On** is **MAC Length/Type** or **Client Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).

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

See also

Bus Search configuration menus (on page 375)

Ethernet serial bus menu (on page 222)

EtherCAT serial bus search configuration menu

Use the EtherCAT Search configuration menu to define conditions to search for and mark on an EtherCAT bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the EtherCAT bus that you want to search.
Mark On	Select the type of information that you want to search.
Start of Frame	Set to **Start of Frame**.
Type	Select the type that you want to search. Available when **Mark On** is **Protocol**. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header**, and **Datagram Header** is **Command**. Available when **Mark On** is **Mailbox**, **Mailbox** is **Mailbox Header**, and **Mailbox Header** is **Type**. Available when **Mark On** is **Mailbox** and **Mailbox** is **Error Reply Service Data**.
Frame Type	Set frame type as **Basic** or **UDP/IP**. Available when **Mark On** is **Protocol**.
Identification	Sets the IP Protocol pattern that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **IP Header**.

Field or control	Description
Source Address, Destination Address	Sets the source and/or destination IP address pattern that you want to search. Tap a **Hex** or **Decimal** 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 **Mark On** is **IP Header**.
Source Port	Sets the UDP header pattern that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **UDP Header**.
Destination, Source	Sets the MAC destination and/or source address pattern that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **MAC Address**.
Tag Control Information	Sets the tag control information pattern that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Tag Control Information**.
Length	Sets the length pattern that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **EtherCAT Header Length**. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header** and **Datagram Header** is **Length**. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Network Variable**, **Network Variable** is **NV Header** and **NV Header** is **Length**. Available when **Mark On** is **Mailbox**, **Mailbox** is **Mailbox Header** and **Mailbox Header** is **Length**.
Datagram	Select the datagram that you want to search. Available when **Mark On** is **Datagram**
Datagram Header	Select the datagram header that you want to search. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header**.

Field or control	Description
Index	Sets the index that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header**, and **Datagram Header** is **Index**. Available when **Mark On** is **Network Variable**, **Network Variable** is **NV Header** and **NV Header** is **Index**.
Address Type	Select the address type as **Device** or **Logical**. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header**, and **Datagram Header** is **Address**.
Position	Sets the device position pattern that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header**, **Datagram Header** is **Address**, **Address Type** is **Device** and **Device Address** is **Position**.
Offset	Sets the offset pattern that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header**, **Datagram Header** is **Address**, **Address Type** is **Device**, and **Device Address** is **Offset**.
Address	Sets the address that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header**, **Datagram Header** is **Address**, and **Address Type** is **Logical**. Available when **Mark On** is **Mailbox**, **Mailbox** is **Mailbox Header**, and **Mailbox Header** is **Address**.
Circulating Frame	Set the circulating frame that you want to search. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header**, and **Datagram Header** is **Circulating Frame**.
IRQ	Sets the IRQ that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Datagram**, **Datagram** is **Datagram Header** and **Datagram Header** is **IRQ**.

Field or control	Description
Data Bytes	Sets the data bytes that you want to search. Available when **Mark On** is **Datagram**, **Datagram** is **Data**. Available when **Mark On** is **Network Variable** and **Network Variable** is **Data**.
Data	Sets the data that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Datagram**, **Datagram** is **Data**. Available when **Mark On** is **Network Variable** and **Network Variable** is **Data**.
Working Counter	Sets the working counter that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Datagram**, **Datagram** is **Working Counter**.
Network Variable	Select the network variable that you want to search. Available when **Mark On** is **Network Variable**.
Publisher ID, NV Count, Cycle Number	Sets the publisher ID, NV count, and cycle number that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Network Variable** and **Network Variable** is **Publisher Header**.
Hash	Sets the hash algorithm that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Network Variable**, **Network Variable** is **NV Header** and **NV Header** is **Hash Algorithm**.
Quality	Sets the quality of the NV header that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Network Variable**, **Network Variable** is **NV Header** and **NV Header** is **Quality**.
Mailbox	Sets the mailbox for which to search. Available when **Mark On** is **Mailbox**.
Channel	Select the channel that you want to search. Available when **Mark On** is **Mailbox**, **Mailbox** is **Mailbox Header**, and **Mailbox Header** is **Channel**.

Field or control	Description
Priority	Select the priority that you want to search. Available when **Mark On** is **Mailbox**, **Mailbox** is **Mailbox Header**, and **Mailbox Header** is **Priority**.
Mailbox Type	Select the mailbox type for which to search. Available when **Mark On** is **Mailbox**, **Mailbox** is **Mailbox Header**, and **Mailbox Header** is **Type**.
Counter	Sets the counter that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Mailbox**, **Mailbox** is **Mailbox Header** and **Mailbox Header** is **Counter**.
Service Data Bytes	Sets the service data bytes that you want to search. Available when **Mark On** is **Mailbox**, **Mailbox** is **Service Data**.
Service Data	Sets the service data that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Mailbox**, **Mailbox** is **Service Data**.
Detail	Sets the detail that you want to search. Available when **Mark On** is **Mailbox**, **Mailbox** is **Error Reply Service Data**, and **Type** is **Detail**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. EtherCAT is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. EtherCAT is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

EtherCAT serial bus configuration menu (on page 223)

eUSB serial bus search configuration menu

Use the eUSB Search configuration menu to define conditions to search for and mark on an eUSB bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.

Field or control	Description
Search Type	Set to **Bus**.
Source	Select the eUSB bus that you want to search.
Mark On	Select the type of information for which to search.
Sync Bits Search	Select to search for the number of sync bits. Available when **Mark On** is **Sync**.
Sync Bits	Set the number of sync bits. By default Sync Bits are OFF. **NOTE:** Sync Bits will be On, only when Speed is High in the Bus configuration. Available when **Mark On** is **Sync**, **Sync Bits Search** is **ON** and **Mark When Sync Bits** is all except, **Inside Range** and **Outside Range**.
Handshake Type	Sets the handshake packet type for which to search. Available options are **Any (XX10)**, **ACK (0010)**, **NAK (1010)**, **STALL (1110)**, or **NYET (0110)**. **NOTE:** NYET is applicable only in High Speed. Available only when **Mark On** is **Handshake Packet**.
Packet Type	Sets the special packet type for which to search. Available options are **Any (XX00)**, **PRE (1100)**, and **RESERVED (0000)**. **NOTE: ERR (1100)**, **SPLIT (1000)**, **PING (0100)** are available only for High Speed. Available only when **Mark On** is **Special Packet**.
Hub Address	Sets the special packet address pattern for which to search. Tap the **Binary** or **Hex** 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 only when **Mark On** is **Special Packet** and **Packet Type** is **Any (XX00)**, **PRE (1100)**, or **RESERVED (0000)**. Available when high speed is selected and **Mark On** is **Special Packet**, **ERR (1100)**, **SPLIT (1000)**, **PING (0100)** are displayed.
Port Address	Sets the special packet address pattern for which to search. Tap the **Binary** or **Hex** 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 only when **Mark On** is **Special Packet** and **Packet Type** is **Any (XX00)**, **PRE (1100)**, or **RESERVED (0000).**. Available when high speed is selected and **Mark On** is **Special Packet**, **ERR (1100)**, **SPLIT (1000)**, **PING (0100)** are displayed.

Field or control	Description
Error Type	Sets the error type for which to search. Available options are **PID Check Bits**, **Token CRC5**, **Data CRC16**, and **Bit Stuffing**. Bit Stuffing is not available when High Speed is selected inthe configuration. Available only when **Mark On** is **Error**.
Token Type	Sets the token packet type for which to search. Available only when **Mark On** is **Token Packet**.
Mark When Address	Sets the token type pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Token Packet**, **Token Type** is all except **SOF (0101)**, **Mark When Address** = **Inside Range**, and **Outside Range**.
Address	Sets the token packet address pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Token Packet** and **Token Type** is all except **SOF (0101)**. When **Mark On** is **Register Access Protocol** and **Fields** is **Address**.
Endpoint	Sets the token packet endpoint pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Token Packet** and **Token Type** = all except **SOF (0101)**.
Frame Number	Sets the frame number pattern for which to search. Tap the **Binary**, **Hex**, or **Decimal** 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 only when **Mark On** is **Token Packet** and **Token Type** is **SOF (0101)**.
Data Packet Type	Sets the data packet type for which to search. Available only when **Mark On** is **Data Packet**. Available options are **ANY (1011)**, **DATA0 (1011)**, and **DATA1 (1011)**. Available options **DATA2 (0111)** and **MDATA (1111)** when High Speed is selected in the configuration.

Field or control	Description
Data Bytes	Sets the number of data bytes for which to search (one to sixteen bytes). Tap the field and use the **A** knob to change the value. Available only when **Mark On** is **Data Packet**.
Byte Offset	Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the **A** knob to change the value. Available only when **Mark On** is **Data Packet**.
Data	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 Mark When field to specify the exact search condition. Tap the **Binary** or **Hex** or **ASCII** 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 only when **Mark On** is **Data Packet** and **Mark On** is **Data Packet** is **Register Access Protocol**, **Fields** is **Data**.
Reset	Select to search all the reset events. It indicates bus reset.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** or **ASCII** 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 only when **Mark On** is **Token Packet** or **Data Packet**.
Resume/Wake	Select to search for all the Resume/Wakeup events.
Connect	Select to search for all the connected events. This event is triggered when the device is connected after POR.
Control Message	Select to search for all the control messages.
Suspend	Select to search all the suspended events.
Device Chirp	Select to search for all the device chirp events.
Host chirp	Select to search for all the host chirp events.
Port Reset	Select to search for all the port reset events.
Port Configuration	Select to search for port configuration events.
Fields	Select the fields. Available options are Command, Address, or Data. Available only when **Mark On** is **Register Access Protocol**.

Field or control	Description
RAP Command	Available only when **Mark On** is **Register Access Protocol** and **Fields** is **Command**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. eUSB is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. eUSB is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

eUSB serial bus configuration menu (on page 224)

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.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the FlexRay bus for which to search.
Mark On	Select the type of information for which to search.
Indicator Bits	Select the defined indicator bits type for which to search from the drop-down list. Available only when **Mark On** is **Indicator Bits**.
Indicator Bits	Enter the indicator bits for which to search. Tap the **Binary**, **Hex**, or **Decimal** 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 only when **Mark On** is **Header Fields**.
Identifier	Enter the frame identifier pattern for which to search. Tap the **Binary**, **Hex**, or **Decimal** 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 only when **Mark On** is **Frame ID** or **Header Fields**.

Field or control	Description
Cycle Count	Enter the cycle count pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Cycle Count** or **Header Fields**.
Payload Length	Enter the payload length pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Header Fields**.
Header CRC	Enter the header CRC pattern for which to search. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Header Fields**.
Data Bytes	Enter the number of data bytes for which to search (one to sixteen bytes). Use the **A** knob to change the value. Available only when **Mark On** is **Data** or **Identifier & Data**.
Data	Enter the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Data** or **Identifier & Data**.
Byte Offset	Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the **A** knob to change the value. Available only when **Mark On** is **Data** or **Identifier & Data**.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Frame ID**, **Cycle Count**, **Data**, or **Identifier & Data**.
Frame Type	Sets the end of frame type for which to search. Available only when **Mark On** is **End of Frame**.

Field or control	Description
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

FlexRay serial bus configuration menu (on page 226)

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.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the I^2C bus that you want to search.
	Select the type of information for which to search.
Direction	Sets the transfer direction for which to search. Available when is **Address** or **Address & Data**.
Addressing Mode	Sets the slave device address length (7 bits or 10 bits long). Available when is **Address** or **Address & Data**.
Address	Sets the address pattern for which to search. The number of bits shown depends on the **Address Mode** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when is **Address** or **Address & Data**.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Use the **A** knob to change the value. Available when is **Data** or **Address & Data**.

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

See also

Bus Search configuration menus (on page 375)

I2C serial bus configuration menu (on page 227)

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.

I3C search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Source	Select the I3C bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates to show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Mark On = Start

This table lists the menu fields displayed when **Mark On** is set to **Start**.

Field or control	Description
Start	Sets the search to mark **Start** events. A **Start** event occurs when the SDA line transitions from a High to Low state while the SCL line remains High. There are no menu fields or controls displayed when this item is selected.

Mark On = Repeated Start

This table lists the menu fields displayed when **Mark On** is set to **Repeated Start**.

Field or control	Description
Repeated Start	Sets the search to mark **Repeated Start** events. A **Repeated Start** is when two or more instances of a **Start** in a row occur without an intervening Stop. There are no menu fields or controls displayed when this item is selected.

Mark On = Address

This table lists the menu fields displayed when **Mark On** is set to **Address**.

Field or control	Description
Direction	Sets the search to mark address events of the specified direction (**Read**, **Write**, or **Either**).
Addressing Mode	Sets the search to mark **7-Bit** or **10-Bit** address events.
Address	Sets the search to mark events that meet the specified address for a Device or register location. The **Addressing Mode** sets the number of bits in the field. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.

Mark On = Data

This table lists the menu fields displayed when **Mark On** is set to **Data**.

Field or control	Description
Data Bytes	Sets the number of data bytes to show in the **Data** field. Valid range is 1-5.
Data	Sets the search to mark events that meet the specified data value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.

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
Enable Slave	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. There are no menu fields or controls displayed when this item is selected.
Disable Slave	Sets the search to mark Disable events (DISEC). Events occur when the Master disables Slave-initiated interrupts on the I3C Bus. There are no menu fields or controls displayed when this item is selected.
Enter Activity State	Sets the search to mark **Enter Activity State** events (ENTAS0, ENTAS1, ENTAS2, ENTAS3). Events occur when the Master sets an activity mode state (0, 1, 2, or 3). There are no menu fields or controls displayed when this item is selected.
Reset Dynamic Address	Sets the search to mark **Reset Dynamic Address** 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. There are no menu fields or controls displayed when this item is selected.
Set Max Write Length	Sets the search to mark **Set Max Write Length** events (SETMWL). Events occur when the Master sets a maximum data write length for a Slave device. There are no menu fields or controls displayed when this item is selected.
Set Max Read Length	Sets the search to mark **Set Max Read Length** events (SETMRL). Events occur when the Master sets a maximum data read length for a Slave device. There are no menu fields or controls displayed when this item is selected.
Set Exchange Time	Sets the search to mark **Set Exchange Time** 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. There are no menu fields or controls displayed when this item is selected.
Set Dynamic Address	Sets the search to mark **Set Dynamic Address** events (SETDASA). Events occur when the Master assigns a dynamic address to one Slave device using the Slave's static address. There are no menu fields or controls displayed when this item is selected.
Set New Dynamic Address	Sets the search to mark **Set New Dynamic Address** events (SETNEWDA). Events occur when the Master assigns a new dynamic address to an I3C Slave device. There are no menu fields or controls displayed when this item is selected.
Get Max Write Length	Sets the search to mark **Get Max Write Length** events (GETMWL). Events occur when the Master gets a maximum data write length in bytes for a slave device. There are no menu fields or controls displayed when this item is selected.

I3C SDR Direct field	Description and menu fields
Get Max Read Length	Sets the search to mark **Get Max Read Length** events (GETMRL). Events occur when the Master gets a maximum data read length in bytes for a slave device. There are no menu fields or controls displayed when this item is selected.
Get Provisional ID	Sets the search to mark **Get Provisional ID** 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. There are no menu fields or controls displayed when this item is selected.
Get Bus Characteristics	Sets the search to mark **Get Bus Characteristics** events (GETBCR). Events occur when the Master sends a Get request to one I3C Slave device to return its Bus Characteristics Register (BCR) contents. There are no menu fields or controls displayed when this item is selected.
Get Dev Characteristics	Sets the search to mark **Get Dev Characteristics** events (GETDCR). Events occur when the Master sends a Get request to one I3C Slave device to return its Device Characteristics Register (DCR) contents. There are no menu fields or controls displayed when this item is selected.
Get Slave Current Status	Sets the search to mark get device status events (GETSTATUS). When the **Packet** button is set to **Direct**, all events are marked. When the **Packet** button is set to **Response**, the menu displays the following fields to refine the search: **Slave Address** sets the Slave's original 7-bit static I^2C address (or 00 if the slave does not have a static I^2C address) for which to search. **Get Status MSB** sets the most significant bits of the Slave's status register for which to search. These 8 bits contain vendor-specific information. **Get Status LSB** 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.
Get Accept Mastership	Sets the search to mark **Get Accept Mastership** events (GETACCMST). Events occur when a Master request occurs, and when the Current Master offers Mastership to an I3C Secondary Master. There are no menu fields or controls displayed when this item is selected.

I3C SDR Direct field	Description and menu fields
Set Bridge Direct Target	Sets the search to mark **Set Bridge Direct Target** 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). When the **Packet** is set to **Direct**, all **Set Bridge Direct Target** events are marked. When the **Packet** is set to **Response**, the menu displays the following fields to refine the search: **Slave Address** sets the Slave's original 7-bit static I2C address, or 00 if the slave does not have a static I2C address. **Dynamic Address** sets the 7-bit slave device address for which to search. **ID** sets the 16-bit unambiguous identifier for the Bridged device, for which to search.
Get Max Data Speed	Sets the search to mark **Get Max Data Speed** 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 (& optionally max. Read Turnaround time) When the **Packet** button is set to **Direct**, all **Get Max Data Speed** events are marked. When the **Packet** button is set to **Response**, the menu displays the following fields to refine the search: **Slave Address** sets the 7-bit address of a specific Slave device for which to search. **Max Write** is an 8-bit field that sets the maximum sustained data write speed (Master to Slave) for which to search. **Max Read** 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. **Max Read Turn** is a 24-bit field specifying the Maximum Read Turnaround Time for which to search.
Get HDR Capability	Sets the search to mark **Get HDR Capability** (GETHDRCAP) events. Events occur when the Master queries a Slave device to determine what HDR Mode(s) that device supports. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.0** in the I3C bus configration menu.

I3C SDR Direct field	Description and menu fields
Get Exchange Time	Sets the search to mark **Get Exchange Time** events (GETXTIME). Use the following fields to refine the search: When the **Packet** button is set to **Direct**, all **Get Exchange Time** events are marked. When the **Packet** button is set to **Response**, the menu displays the following fields to refine the search: **Slave Address** sets the 7-bit address of a specific Slave device for which to search. **Supported Modes Byte** is an 8-bit field that sets which Timing Control Mode(s) to search for on the specified Slave device. **State Byte** 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. **Frequency Byte** is an 8-bit field that sets the Slave's internal oscillator frequency for which to search. **Inaccuracy Byte** is an 8-bit field that sets the maximum variation of the Slave's internal oscillator for which to search.
Get Optional Feature Capabilities	Sets the search to mark **Get Optional Feature Capabilities** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Data Transfer Ending Procedure Control	Sets the search to mark **Data Transfer Ending Procedure Control** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Set Group Address	Sets the search to mark **Set Group Address** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Slave Reset Action	Sets the search to mark **Slave Reset Action** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Reset Group Address	Sets the search to mark **Reset Group Address** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.

I3C SDR Direct field	Description and menu fields
Multi-Lane Data Transfer Control	Sets the search to mark **Multi-Lane Data Transfer Control** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Set Route	Sets the search to mark **Set Route** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Device to Device(s) Tunneling Control	Sets the search to mark **Device to Device(s) Tunneling Control** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.

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
Enable Slave	Sets the search to mark **Enable Slave** 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. Use the following field to refine the search: **Event Byte** sets the Slave enable value for which to search.
Disable Slave	Sets the search to mark **Disable Slave** 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. Use the following field to refine the search: **Event Byte** sets the Slave disable value for which to search.
Enter Activity State	Sets the search to mark **Enter Activity State** events (ENTAS0, ENTAS1, ENTAS2, ENTAS3). Events occur when the Master sets an activity mode state (0, 1, 2, or 3). There are no menu fields or controls displayed when this item is selected.
Reset Dynamic Address	Sets the search to mark **Reset Dynamic Address** events (RSTDAA). Events occur when the Master clears/resets one or all Master-assigned Dynamic addresses from Slave devices. There are no menu fields or controls displayed when this item is selected.

I3C SDR Broadcast field	Description and menu fields
Enter Dynamic Address	Sets the search to mark **Enter Dynamic Address** events (ENTDAA). Events occur when the Master requests Slave devices to enter the Dynamic Address Assignment process. When enter dynamic address is set **Packet** field gets enabled. Sets the Packet as **Response** or **Broadcast**. When the **Packet** is set to **Broadcast**, all **Enter Dynamic Address** events are marked. When the **Packet** is set to **Response**, the menu displays the following fields to refine the search: **Slave Address** sets the 7-bit address of a Slave device for which to search. **ID** sets the Slave ID for which to search. **BCR** sets the Bus Control Register values for which to search. **DCR** sets the Device Control Register values for which to search.
Set Max Write Length	Sets the search to mark **Set Max Write Length** events (SETMWL). Events occur when the Master sets a maximum data write length in bytes for a slave device. There are no menu fields or controls displayed when this item is selected.
Set Max Read Length	Sets the search to mark **Set Max Read Length** events (SETMRL). Events occur when the Master sets a maximum data read length in bytes for a slave device. There are no menu fields or controls displayed when this item is selected.
Define List of Slaves	Sets the search to mark **Define List of Slaves** 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. Use the following fields to refine the search: **Dynamic Address** sets the devices' dynamic 7-bit address for which to search. **DCR Type** sets the Device Control Register values for which to search. **BCR Type** sets the Bus Control Register values for which to search. **Static Address** sets the devices' 7-bit static address for which to search.
Enter Test Mode	Sets the search to mark **Enter Test Mode** events (ENTTM). Events occur when all I3C devices are told that the Master is entering a specified Test Mode during manufacturing or Device test. Use the following field to refine the search: **Test Mode** sets the text mode value for which to search.
Set Exchange Time	Sets the search to mark **Set Exchange Timing Information** 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. There are no menu fields or controls displayed when this item is selected.

I3C SDR Broadcast field	Description and menu fields
Set Static as Dynamic Address	Sets the search to mark events where the Master assigns a Dynamic address to a Slave with a known Static address. There are no menu fields or controls displayed when this item is selected.
Data Transfer Ending Procedure Control	Sets the search to mark **Data Transfer Ending Procedure Control** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Set Group Address	Sets the search to mark **Set Group Address** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Slave Reset Action	Sets the search to mark **Slave Reset Action** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Define List of Group Addresses	Sets the search to mark **Define List of Group Addresses** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Reset Group Address	Sets the search to mark **Reset Group Address** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Multi-Lane Data Transfer Control	Sets the search to mark **Multi-Lane Data Transfer Control** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Set Bus Context	Sets the search to mark **Set Bus Context** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.

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
I3C DDR	Sets the search to mark Double Data Rate (DDR) events. Use the following fields to refine the search: **Slave Address** sets the 7-bit address of a Slave device for which to search. **Command Code** sets the read/write values of a command word for which to search. **Data Words** sets the length of the data words for which to search, in the range of integer value 1 (16 bits) to 5 (80 bits). **Data** sets the data value for which to search.

Mark On = Errors fields

The following table lists the fields available when **Mark On** is set to **Errors**.

Errors field	Description
Missing Ack	Sets the search to mark missing ACK error events. There are no menu fields or controls displayed when this item is selected.
T-Bit	Sets the search to mark **T-Bit** (transition bit) error events. T-bits are an alternative to the ACK/NACK mechanism. There are no menu fields or controls displayed when this item is selected.
Parity	Sets the search to mark **Parity** error events. There are no menu fields or controls displayed when this item is selected.
Broadcast Address	Sets the search to mark **Broadcast Address** error events (an error with Broadcast Address commands intended for multiple Slave devices). There are no menu fields or controls displayed when this item is selected.
Dynamic Address	Sets the search to mark **Dynamic Address** events. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.
Preamble	Sets the search to mark **Preamble** error events. The Preamble is the set of bits preceding the data words in HDR-DDR. There are no menu fields or controls displayed when this item is selected.
CRC5	Sets the search to mark **CRC5** (Cyclic Redundancy Check, with fifth-order polynomial length) error events. There are no menu fields or controls displayed when this item is selected.

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
Hot-Join	Sets the search to mark **Hot-Join** 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
Direct Message End	Sets the search to mark **Direct Message End** 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
Stop	Sets the search to mark **Stop** 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
HDR Restart	Sets the search to mark **HDR Restart** events. An event occurs when an **HDR Restart** 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
HDR Exit	Sets the search to mark **HDR Exit** events. An event occurs when an **HDR Exit** pattern is sent on the bus. There are no menu fields or controls displayed when this item is selected.

Mark On = Slave Reset Pattern

The following table lists the fields available when **Mark On** is set to **Slave Reset Pattern**.

Slave Reset Pattern field	Description and menu fields
Slave Reset Pattern	Sets the search to mark **Slave Reset Pattern** events. An event occurs when an **Slave Reset Pattern** pattern is sent on the bus. There are no menu fields or controls displayed when this item is selected. This item is available when **Version** is **1.1** in the I3C bus configration menu.

See also

Bus Search configuration menus (on page 375)

I3C serial bus configuration menu (on page 228)

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.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the LIN bus that you want to search.
Mark On	Select the type of information for which to search.
Identifier	Sets the identifier pattern for which to search. Tap the **Binary**, **Hex**, or **Decimal** 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 only when **Mark On** is **Identifier** or **Identifier & Data**.
Mark When	Sets the mark when condition. When set to **Inside Range** or **Outside Range**, fields are displayed to set a **Data Low** and **Data High** boundary pattern for the specified search range. Available only when **Mark On** is **Data** or **Identifier & Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Data** or **Identifier & Data**.
Data Bytes	Sets the number of data bytes for which to search (one to four bytes). Use the **A** knob to change the value. Available only when **Mark On** is **Data** or **Identifier & Data**.

Field or control	Description
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

LIN serial bus configuration menu (on page 229)

Manchester serial bus search configuration menu

Use the Manchester Search configuration menu to define conditions to search for and mark on an manchester bus decode. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the Manchester bus that you want to search.
Mark On	Select the type of information for which to search.
Sync	Sets the synchronization type for which to search. Tap the **Binary** or **Hex** 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. Only available when **Mark On** is **Sync Bits**.
Header	Sets the header for which to search. Tap the **Binary** or **Hex** 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. Only available when **Mark On** is **Header**.
Data Words	Set the length of the data words for which to search, in the range of integer value Default value is 1, maximum value is 5. Only available when **Mark On** is **Data**.
Data	Sets the data type for which to search. Tap the **Binary** or **Hex** 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. Only available when **Mark On** = **Data**.

Field or control	Description
Trailer	Sets the trailer for which to search. Tap the **Binary** or **Hex** 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. Only available when **Mark On** is **Trailer**.
Error Type	Sets the error type for which to search. Available types are **Manchester** and **Parity**. Only available when **Mark On** is **Errors**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. Manchester is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. Manchester is currently not supported as a bus trigger source.

NOTE: In the bus configuration, when the packet view is

- **On**, the displayed Search options are **Sync Bits**, **Header**, **Data**, **Trailer**, and **Errors**.
- **Off**, the displayed Search options are **Data** and **Errors**.

See also

Bus Search configuration menus (on page 375)

Manchester serial bus configuration menu (on page 230)

MDIO serial bus search configuration menu

Use the MDIO Search configuration menu to define the conditions to search for and mark on an MDIO bus waveform. You can have multiple searches on the same bus.

MDIO search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Search Type	Set to **Bus**.
Source	Select the MDIO bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates to show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. MDIO is currently not supported as a bus trigger source.

Field or control	Description
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. MDIO is currently not supported as a bus trigger source.

Mark On = Start Packet

This table lists the menu fields displayed when **Mark On** is set to **Start Packet**.

Field or control	Description
Start Packet	Sets the search to mark **Start** events. A **Start Packet** event occurs when the SDA line transitions from a High to Low state while the SCL line remains High. There are no menu fields or controls displayed when this item is selected.

Mark On = OpCode

This table lists the menu fields displayed when **Mark On** is set to **OpCode**.

Field or control	Description
OpCode	Sets to search two bit operation codes. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark When** is **OpCode**.

Mark On = Physical Address

This table lists the menu fields displayed when **Mark On** is set to **Physical Address**.

Field or control	Description
Physical Address	Set the 5 bit long field to search for the physical PHYs which are used to interface to the network. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.

Mark On = Register Address

This table lists the menu fields displayed when **Mark On** is set to **Register Address**.

Field or control	Description
Register Address	Sets the 5 bit long field to search the register to be written to or read from. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Register Address**.

Mark On = Device Type

This table lists the menu fields displayed when **Mark On** is set to **Device Type**.

Field or control	Description
Device Type	Sets the 5 bit long field to search for the device type of clause 45. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Device Type**.

Mark On = Data

This table lists the menu fields displayed when **Mark On** is set to **Data**.

Field or control	Description
Data	Sets the search to mark events that meet the specified data value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Data**.

Mark On = Address

This table lists the menu fields displayed when **Mark On** is set to **Address**.

Field or control	Description
Address	Sets the 16 bit long field search for the specified address. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** = **Address**.

Mark On = Error

This table lists the menu fields displayed when **Mark On** is set to **Error**.

Field or control	Description
Error	Sets the error on which to search. Available when **Mark On** is **Error**.
Error Type	Sets the error type on which to search. Select the error type from the drop-down. Available error types are: ▪ OpCode ▪ Device Type ▪ Any Available when **Mark On** is **Error**.

See also

[Bus Search configuration menus](#) (on page 375)

[MDIO serial bus configuration menu](#) (on page 231)

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 MIL-STD-1553 bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the MIL-STD-1553 bus that you want to search.
Mark On	Select the type of information for which to search.
Transmit/Receive Bit	Sets the transmit or receive bit state for which to search. Only available when **Mark On** is **Command**.
Mark when RT Address	Sets the RT address condition for which to search. When set to **Inside Range** or **Outside Range**, fields are displayed to set a low and high address for the specified search range. Only available when **Mark On** is **Command**.
Parity	Sets the parity state for which to search. Only available when **Mark On** is **Command**.
Address	Sets the address value for which to search. Tap the **Binary**, **Hex**, or **Decimal** 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. Only available when **Mark On** is **Command**.
Low Address	Sets the low address value for which to search. Tap the **Binary**, **Hex**, or **Decimal** 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. Only available when **Mark On** is **Command** and **Mark When RT Address** is **Inside Range** or **Outside Range**.
Subaddress/Mode	Sets the subaddress or mode value for which to search. Tap the **Binary**, **Hex**, or **Decimal** 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. Only available when **Mark On** is **Command** and **Mark When RT Address** is **Inside Range** or **Outside Range**.

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

See also

Bus Search configuration menus (on page 375)

MIL-STD-1553 serial bus menu (on page 232)

NFC serial bus search configuration menu

Use the NFC Search configuration menu to define conditions to search for and mark on an NFC bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying the search marks on this search.

Field or control	Description
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the **NFC** bus that you want to search.
Mark On	Select the type of information that you want to search. The mark on menu changes depending on the Standard selcted in the NFC bus configuration menu.
Type	Set the type of packet to search on as **Command** or **Response**.
Data Bytes	Sets the number of data bytes to search (one to five bytes). Use the A knob to change the value. The data bytes field is available for the Standard 15693 and 14443B in the NFC bus configuration menu. Available when **Mark On** is **Data** in the NFC search configuration menu.
Data	Sets the data pattern to search. The number of bits shown depends on the Data Bytes setting. Tap the Binary or Hex field and use the A and B knobs to select the values. Or double-tap on the field and use the virtual keypad to enter the values. The data field is available for the Standard 15693 and 14443B in the NFC bus configuration menu. Available when **Mark On** is **Data** in the NFC search configuration menu.
UID	Sets the UID that you want to search. Tap a Binary or Hex field and use the A and B knobs to select the values. Or double-tap on the field and use the virtual keypad to enter the values. The UID field is available for the Standard 15693 and 14443A in the NFC bus configuration menu. Available when **Mark On** is **UID**.
Command Codes	Sets the command codes to search. The command codes menu changes depending on the Standard selected in the NFC bus configuration menu. Available when **Mark On** is **Command code** and **Type** is **Command**.
Response Codes	Sets the response codes. The response codes field is available for the Standard 14443A, 14443B, and FeliCa in the NFC bus configuration menu. The response codes menu changes depending on the Standard selected in the NFC bus configuration menu. Available when **Mark On** is **Response code** and **Type** is **Response**.
Error Type	Sets the error bytes that you want to search. The error type is CRC and cannot be edited for the Standards 15693, 14443B, and FeliCa. For Standard 14443A, set the type as Any, Parity, or CRC. Available when **Mark On** is **Error**.

Field or control	Description
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. NFC is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. NFC is currently not supported as a bus trigger source.

See also

Bus Trigger configuration

Bus Search configuration menus (on page 375)

NFC serial bus configuration menu (on page 234)

NRZ serial bus search configuration menu

Use the NRZ Search configuration menu to define conditions to search for and mark on a NRZ bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Source	Select the NRZ bus that you want to search.
Mark On	Select the type of information for which to search.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Use the **A** knob to change the value. Available when **Mark On** is **Data**. Available when **Packet View** is **Off** from the NRZ bus configuration menu.
Data Words	Sets the number of data bytes to search for (one to five bytes). Use the A knob to change the value. Available when **Mark On** is **Data**. Available when **Packet View** is **On** from the NRZ bus configuration menu.
Data	Sets the data pattern to search for. The number of bits shown depends on the **Data Bytes** or **Data Words** setting. Available when **Mark On** is **Data**. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
Error Type	Sets the error type as parity to search for. Available when **Mark On** is **Errors**. Available when **Packet View** is **On** from the NRZ bus configuration menu.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. NRZ is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. NRZ is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

NRZ serial bus configuration menu (on page 235)

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
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the parallel bus that you want to search.
Data	Sets the data pattern for which to search. The number of bits shown depends on how the parallel bus is defined. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

Parallel Bus configuration menu (on page 237)

PCIe serial bus search configuration menu

Use the PCIe Search configuration menu to define conditions to search for and mark on PCIe bus waveform. You can have multiple searches on the same bus.

Note that the fields and controls changes based on the selected fields from PCIe bus configuration menu.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Search Type	Set to **Bus**.

Field or control	Description
Source	Select the PCIe bus that you want to search.
Mark On	Select the type of information for which to search. When **Packet View** is set to **On** from PCIe bus configuration menu, the available options are: ▪ For **2.5 Gb/s** or **5 Gb/s** speed signal: **Ordered Set**, **Control Character**, **DLLP**, **TLP**, and **Errors** ▪ For **8 Gb/s** or **16 Gb/s** speed signal: **Ordered Set**, **DLLP**, **TLP**, and **Errors** When **Packet View** is set to **Off** from PCIe bus configuration menu, the available options are : ▪ For **2.5 Gb/s** or **5 Gb/s** speed signal: **Ordered Set, Custom Pattern**, **Control Character**, and **Errors** ▪ For **8 Gb/s** or **16 Gb/s** speed signal: **Sync Header**, **Compliance Pattern**, and **Ordered Set**
Character	Sets the control character for which to search. Available when **Mark On** is set to **Control character** . Available options are **COM (K28.5)**, **EDB (K30.7)**, **END (K29.7)**, **FTS(K28.1)**, **IDL (K28.3)**, **PAD (K23.7)**, **SDP (K28.2)**, **SKP (K28.0)**, **STP (K27.7)**, **EIE (K28.7)**, and **Any**.
DLLP Type	Sets the DLL packet for which to search. Available options are **ACK/NAK**, **Power Management**, **Flow Control**, **Vendor-Specific**, **NOP**, **Data Link Feature**, and **MR IOV**. Available when **Mark On** is set to **DLLP**.
DLLP	Sets the DLLP for which to search. Available when **Mark On** is set to **DLLP**. Available when **Mark On** is set to **DLLP**. When **DLLP Type** is set to **ACK/NAK**, the availble optoins are **Any**, **ACK**, and **NAK**. When **DLLP Type** is set to **Power Management**, the availble optoins are **PM_Enter_L1**, **PM_Enter_L23**, **PM_Active_State_...**, and **PM_Request_Ack**. When **DLLP Type** is set to **Flow Control**, the availble optoins are **InitFC1-P**, **InitFC1-NP**, **InitFC1-Cpl**, **InitFC2-P**, **InitFC2-NP**, **InitFC2-Cpl**, **UpdateFC-P**, **UpdateFC-NP**, and **UpdateFC-Cpl**. When **DLLP Type** is set to **MR IOV**, the available options are **MRInit**, **MRInitFC1**, **MRInitFC2**, and **MRUpdateFC**.
Specify	Select the specify as **Virtual Channel**, **Header Credits**, and/or **Data Credits**. Available when **Mark On** is set to **DLLP**, **DLLP Type** is **Flow Control**.
Qualifier	Select the qualifier for which to search. Available when **Mark On** is set to **DLLP**, **DLLP Type** is **Flow Control** and **Specify** is **Vitual Channel**, **Header Credits** and/or **Data Credits**. Available options are **=**, **≠**, **>**, **<**, and **>**.

Field or control	Description
Value	Sets a specfic **Specify** value for which to search. Available when **Mark On** is set to **DLLP**, **DLLP Type** is **Flow Control**. Use knob **A** to set a value, or double-tap the field and enter a value using the virtual keyboard.
TLP Type	Sets the Transaction Layer Packet (TLP) type for which to search. Available when **Mark On** is set to **TLP**. Available options are: ▪ For **2.5 Gb/s** speed signal: **Any**, **Memory**, **I/O**, **Config**, **Message**, and **Completion** ▪ For **5 Gb/s**, **8 Gb/s**, or **16 Gb/s** speed signal: **Any**, **Memory**, **I/O**, **Config**, **Message**, **Completion**, **Atomic Operation**, and **Prefix**
Prefix Type	Sets the prefix type for which to search **Any**, **Local**, or **End-to-End**. Available when **Mark On** is in **TLP** and **TLP Type** is set to **Prefix**.
Sequence Number	Sets the sequence number pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is in **DLLP** and **DLLP Type** is set to **ACK/NAK**. Available when **Mark On** is in **TLP** and **TLP Type** is set to **Any**, **Memory**, **I/O**, **Config**, **Message**, **Completion**, **Prefix**, or **Atomic Operation**.
Requester ID	Sets the requester ID pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is in **TLP** and **TLP Type** is set to **Any**, **Memory**, **I/O**, **Config**, **Message**, **Completion**, or **Atomic Operation**.
Prefix	Sets the prefix pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
Steering Tag	Sets the steering tag pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
Completer ID	Sets the completer ID pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is in **TLP** and **TLP Type** is set to **Completion**.
Bus Number	Sets the bus number pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is in **TLP** and **TLP Type** is set to **Config** or **Message**.

Field or control	Description
Addressing Mode	Sets the address length as **32-Bit** or **64-Bit**. Available when **Mark On** is in **TLP** and **TLP Type** is set to **Memory** or **Atomic Operation**.
Address High/Address Low	Sets the boundary address conditions for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is in **TLP** and **TLP Type** is set to **Memory**, **Message**, or **Atomic Operation**.
Message Code	Sets the message code pattern for which to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is in **TLP** and **TLP Type** is set to **Message**.
Direction	Sets the search to mark address events of the specified direction **Either**, **Read**, or **Write**. Available when **Mark On** is in **TLP** and **TLP Type** is set to **I/O**.
Address	Sets the address that you want to search. Available when **Mark On** is in **TLP** and **TLP Type** is set to **I/O**. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
Operations	Sets the operation for which to search. Available when **Mark On** is in **TLP** and **TLP Type** is set to **Atomic Operations**. Available options are **Any**, **Fetch Add**, **Swap**, and **CAS**.
Sync Header	Sets the sync header for which to search. Available when **Mark On** is set to **Sync Header** and the fileds are **Ordered Set (01b)**, **Data (10b)**, and **Invalid (00b or 11b)**. Available when **Mark On** is set to **Custom Pattern** and the fileds are **Control**, **Data**, and **Either**.
Ordered Set	Sets the ordered set for which to search. Availabe when **Mark On** is set to **Ordered Set**. Available options are: ▪ For **2.5 Gb/s** speed signal: **TS1**, **TS2**, **EIOS**, **FTS**, and **SKP** ▪ For **5 Gb/s** speed signal: **TS1**, **TS2**, **EIOS**, **EIEOS**, **FTS**, and **SKP** ▪ For **8 Gb/s** speed signal: **TS1**, **TS2**, **EIOS**, **EIEOS**, **FTS**, **SDS**, and **SKP** ▪ For **16 Gb/s** speed signal: **TS1**, **TS2**, **EIOS**, **EIEOS**, **FTS**, **SDS**, **SKP**, and **Control SKP**

Field or control	Description
Pattern	Sets the pattern for which to search. Available when **Mark On** is in **Custom Pattern**. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
Symbols	Sets the symbols (length of the pattern between 1 to 16) for which to search. Available when **Mark On** is in **Custom Pattern**.
Error Type	Sets the error type for which to search. Available when **Mark On** is set to **Errors**. Available options are: ▪ For **2.5 Gb/s** or **5 Gb/s**: **CRC**, **LCRC**, **ECRC**, **Frame Length**, **Disparity**, **Symbol**, and **Packet** ▪ For **8 Gb/s** or **16 Gb/s**: **CRC**, **LCRC**, **ECRC**, and **Packet**
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

PCIe serial bus configuration menu (on page 238)

PSI5 serial bus search configuration menu

Use the PSI5 Search configuration menu to define conditions to search for and mark on an PSI5 bus waveform. You can have multiple searches on the same bus.

PSI5 search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Search Type	Set to **Bus**.
Source	Select the PSI5 bus that you want to search.

Field or control	Description
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu updates to show additional fields. See the following **Mark On** tables for the menu fields associated with specific **Mark On** settings. When **Direction** is **ECU to Sensor**, the available settings are: ▪ Start ▪ Data ▪ Function Code ▪ Sensor Address ▪ Register Address ▪ CRC Error When **Direction** is **Sensor to ECU**, the available settings are: ▪ Start ▪ Status Available when the status bit is configured in bus configuration. ▪ Data ▪ Block ID ▪ Sensor Status ▪ Error
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. PSI5 is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. PSI5 is currently not supported as a bus trigger source.

Mark On = Start

This table lists the menu fields displayed when **Mark On** is set to **Start**.

Field or control	Description
Start	Sets the search to mark **Start** events. A **Start** event occurs with two zeros when direction is from sensor to ECU. When direction is from ECU to sensor, Start event occurs with 011111110 when sync mode is pulse width, and 010 when sync mode is tooth gap. When direction is from Sensor to ECU, Start event occurs with 00. There are no menu fields or controls displayed when this item is selected. When **Direction = ECU to Sensor** and **Direction = ECU to Sensor**.

Mark On = Data

This table lists the menu fields displayed when **Mark On** is set to **Data**.

Field or control	Description
Data Bits	When sync mode is tooth gap, data bits are 4 and 8 bits. When sync mode is pulse width data bits are 4 , 8 and 20
Data	Sets the search to mark events that meet the specified data value. For Sensor to ECU, it displays Region A. If the Data B field is configured Bus configuration, displays Region B as data search option. For ECU to Sensor, data search displays Data Bits and Value. In configuration Sync mode is Tooth Gap, the number of data bits is 4 and 8. In configuration Sync Mode is Pulse Width, the data bits are 4, 8, and 20. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Data** and **Direction = ECU to Sensor**.

On = Function Code

This table lists the menu fields displayed when **Mark On** is set to **Function Code**.

Field or control	Description
Function Code	Sets the search to mark events that meet the specified data value. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Function Code** and **Direction = ECU to Sensor**.

Mark On = Sensor Address

This table lists the menu fields displayed when **Mark On** is set to **Sensor Address**.

Field or control	Description
Address	Sets the sensor address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Sensor Address** and **Direction = ECU to Sensor**.

Mark On = Register Address

This table lists the menu fields displayed when **Mark On** is set to **Register Address**.

Field or control	Description
No. of Bits	▪ 2-Bit ▪ 6-Bit ▪ 8-Bit Only available when **Mark On = Register Address** and **Direction = ECU to Sensor**.

Field or control	Description
Register Address	Sets the register address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Register Address** and **Direction = ECU to Sensor**.

Mark On = CRC Error

This table lists the menu fields displayed when **Mark On** is set to **CRC Error**.

Field or control	Description
CRC Error	Sets the search to mark **CRC Error** events. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = CRC Error** and **Direction = ECU to Sensor**.

Mark On = Sensor Status

This table lists the menu fields displayed when **Mark On** is set to **Sensor Status**.

Field or control	Description
Type	Sets the search to mark events that meet the specified type. Available options are **Sensor Ready**, **Sensor Busy**, **Sensor Defect**, **Sensor Unlocked**, and **Service Mode**. Only available when **Mark On = Sensor Status** and **Direction = Sensor to ECU**.

Mark On = Block ID

This table lists the menu fields displayed when **Mark On** is set to **Block ID**.

Field or control	Description
CRC Error	Sets the search to mark **Block ID** events. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Block ID** and **Direction = Sensor to ECU**.

Mark On = Status

This table lists the menu fields displayed when **Mark On** is set to **Status**.

Field or control	Description
CRC Error	Sets the search to mark **Status** events. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Only available when **Mark On = Status** and **Direction = Sensor to ECU**.

Mark On = Error

This table lists the menu fields displayed when **Mark On** is set to **Error**.

Field or control	Description
Error	Sets the search to mark **Error** type. Available error types are **Any**, **Parity**, **CRC**, and **Response Code**. Only available when **Mark On = Error** and **Direction = Sensor to ECU**.

See also

Bus Search configuration menus (on page 375)

PSI5 serial bus configuration menu (on page 240)

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.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the RS232 bus that you want to search.
	Select the type of information for which to search.
Data Bytes	Sets the number of data bytes (1 byte = 8 bits) for which to search (one to ten bytes). Use the **A** knob to change the value. Available when is **Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Words** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when is **Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).

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

See also

Bus Search configuration menus (on page 375)

RS232 serial bus menu (on page 241)

SDLC serial bus search configuration menu

Use SDLC search bus configuration to operate on point-to-point, multipoint or loop facilities on switched/dedicated, two/four-wire circuits with full-duplex or half-duplex operation.

Field	Description
Display	Turns On or Off. Displays the decode bus on the waveform view, when the **Display** is set to **On**.
Search Type	Set to **Bus**.
Source	Select the Bus 1 (SDLC) from the drop-down.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.
Mark On	Sets the type of information to search and mark on the bus waveform acquisition.
Start	Start is a flag event, which starts the transmission frame. Available when **Mark On** is **Start**.
Data	Sets the data pattern. Tap the **Binary** and **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Data**.
Data Bytes	Sets the number of client data bytes (one to five bytes). Use the **A** knob to change the value. Available when **Mark On** is **Data**.
Address	Sets the address value. Tap the Binary and Hex field, 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 the values. Available when **Mark On** is **Address**. Default selection is **Broadcast Address** .

Field	Description
Broadcast	Sets the broadcast address (also called as all-stations address). An address field of all 1's is reserved for the broadcast address only. Available when **Mark On** is **Address**.
No Station	Sets an all 0's address field and is reserved as a "no station's address". Available when **Mark On** is **Address**.
Station Address	Sets its own individual address; a station address. Available when **Mark On** is **Address**.
Type	Search for various frames. Additionally **Unnumbered** frame has **Type** to set **Command**, **Response**, and **Both**. Available when **Mark On** is **Unnumbered**.
Unnumbered Poll	Search an optional response poll is sent by the primary station to poll one station, a group of stations, or all the stations on the loop. Available when **Mark On** is **Unnumbered** and **Type** is **Command**.
Set Normal Response Mode	This command place the secondary station in Normal Response Mode (NRM) for information transfer. Available when **Mark On** is **Unnumbered** and **Type** is **Command**.
Set Normal Response Mode Extended	This command place the secondary station in Normal Response Mode (NRM) or Normal Response Mode Extended (NRME) for information transfer. Available when **Mark On** is **Unnumbered** and **Type** is **Command**.
Unnumbered Acknowledgment	This is the affirmative response to **Set Normal Response Mode**, **Disconnect Mode** or **Set Initialisation Mode** command. Available when **Mark On** is **Unnumbered** and **Type** is **Response**.
Disconnect Mode	Terminates other modes and place the receiving (secondary) station to disconnected mode. Available when **Mark On** is **Unnumbered** and **Type** is **Response**.
Frame Reject	This response is transmitted by a secondary station in normal response mode, when it receives an invalid frame. Available when **Mark On** is **Unnumbered** and **Type** is **Response**.
Beacon	Beacon trasmitts the response, when the secondary station detects the loss of communication at its input. Available when **Mark On** is **Unnumbered** and **Type** is **Response**.

Field	Description
Unnumbered information	As a command or a response is used for transmitting unnumbered information. Available when **Mark On** is **Unnumbered** and **Type** is **Both**.
Set or Request Initialisation Mode	This command initiates system specified procedures for initialising link-level functions. Available when **Mark On** is **Unnumbered** and **Type** is **Both**.
Disconnect or Request Disconnect	This request is sent by a secondary station desired to be disconnected (by the disconnect command). Available when **Mark On** is **Unnumbered** and **Type** is **Both**.
Disconnect Mode	This response is transmitted by the secondary station to indicate that it is in the disconnected mode. Available when **Mark On** is **Unnumbered** and **Type** is **Both**.
Test	As a command, a test frame may sent to a secondary station in any mode to solicit a **Test** response. Available when **Mark On** is **Unnumbered** and **Type** is **Both**.
Exchange station Identification	As a command, **Exchange station Identification** solicits the identification of the receiving (secondary) station. An information field may be included in the frame to identify the transmitting (primary) station. Available when **Mark On** is **Unnumbered** and **Type** is **Both**.
Configure	The configure command contains a function descriptor (a subcommand) in a single-byte information field. Available when **Mark On** is **Unnumbered** and **Type** is **Both**.
Receive Ready Frame	Sent by either a primary or secondary station, **Receive Ready Frame** confirms numbered frames through Nr-1 and indicates that the originating station is ready to receive. Available when **Mark On** is **Supervisory**.
Receive Not Ready Frame	Sent by either a primary or secondary station, **Receive Not Ready Frame** indicates a temporarily busy condition due to buffering or other internal constraints. Available when **Mark On** is **Supervisory**.
Reject Frame	This command/response may transmitted either to request transmission or retransmission of numbered information frames. **Reject Frame** confirms frames through Nr-1 and requests the retransmission of numbered information frames starting at Nr contained in the **Reject Frame**. Available when **Mark On** is **Supervisory**.
FCS	FCS (Field Check sequence) used for Cyclic redundancy check. Available when **Mark On** is **Errors**.

Field	Description
Out of Numeric Order	Search **Reject Frame** status field for Z=1. Available when **Mark On** is **Errors**.
End	End is a flag event, which ends the transmission frame. Available when **Mark On** is **End**.

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.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the **SENT** bus that you want to search.
Mark On	Select the type of information for which to search.
Status / Communications	Sets the value of the status/communications nibble for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Fast Channel**.
Fast Channel 1	Sets the condition and value of the fast channel 1 data for which to search. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤). The default is =. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Fast Channel**.
Fast Channel 2	Sets the condition and value of the fast channel 2 data for which to search. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤). The default is =. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Fast Channel**.
Counter	Sets the condition and value of the counter data for which to search. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤). The default is =. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Fast Channel**.

Field or control	Description
Inverted Nibble	Sets the value of the inverted nibble data for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Fast Channel**.
Message ID	Sets the value of the message ID data for which to search. vailable when **Mark On** is **Slow Channel**.
Data	Sets the condition and value of the slow channel data for which to search. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤). The default is =. Tap the **Binary** or **Hex** field and use the A and B knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Slow Channel**.
Mark When	Sets the mark when condition. Tap the down arrow and select the condition for which to search (=, ≠, >, <, ≥, ≤, **Inside Range**, **Outside Range**). The default is =. When set to **Inside Range** or **Outside Range**, fields are displayed to set high and low boundary values for the number of clock ticks for which to mark. Available when **Mark On** is **Pause Pulse**.
Number of Ticks	Sets the number of Pause Pulse ticks for which to search. Tap the **Ticks High** or **Tick Low** field and use the **A** and **B** knobs to set the values. Available when **Mark On** is **Pause Pulse** and **Mark When** is set to =, ≠, >, <, ≥, or ≤.
Ticks High, **Ticks Low**	Sets the highest and lowest values for the range of Pause Pulse ticks for which to search. Tap the **Ticks High** or **Tick Low** field and use the **A** and **B** knobs to set the values. Available when **Mark On** is **Pause Pulse** and **Mark When** is **Inside Range** or **Outside Range**.
Error Type	Sets the error type for which to search. Tap the arrow and select the error condition. Available when **Mark On** is **Error**.
CRC Type	Sets the CRC error type for which to search. Available when **Mark On** is **Error** and **Error Type** is **CRC**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

SENT serial bus configuration menu (on page 243)

SMBus serial bus search configuration menu

Use the SMBus Search configuration menu to define conditions to search for and mark on an SMBus bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying the search marks on this search.
Search Type	Set to **Bus**.
Source	Select the **SMBus** bus that you want to search.
Mark On	Select the type of information that you want to search.
Start	Select to search on the start events.
Repeated Start	Select to search on the repeated start events.
Address	Sets the 7-bit address pattern that you want to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Address**
Host Address	Select to search on the host address. Available only when **Mark On** is **Host Address** .
Device Address	Set the 7-bit device address that you want to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Device Address**
Command Code	Sets the 8-bit command code that you want to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Command Code**
Data	Sets the data pattern that you want to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Data**.
Data Bytes	Sets the number of data bytes that you want to search (one to eight bytes). Use the **A** knob to change the value. Available only when **Mark On** is **Data**.
Field Bytes	Sets the field bytes as 1, 2, or 4. Available only when **Mark On** is **UDID Data**.

Field or control	Description
UDID Data	Sets the UDID data that you want to search. Tap a **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **UDID Data**.
Error Type	Sets the error bytes that you want to search. You can search on ANY, ACK, NACK, and PEC errors. Available only when **Mark On** is **Error**. PEC error search is available only when the **PEC Byte** in SMbus bus configuration is set as True.
Stop	Select to search on the stop events.
Idle	Select to search on the idle events.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. SMBus is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. SMBus is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

SMBus serial bus configuration menu (on page 244)

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.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the SPI bus that you want to search.
Mark On	Select the type of information for which to search.
Search Source	Set the search source. Set either MOSI or MISO. Available when **Mark On** is **SS Active**.

Field or control	Description
Data Words	Sets the number of data words defined based on the word size in bus configuration. If the word size in bus configuration is 4, then data words will go to 32. Maximum data words is word size <=128. Available when **Mark On** is **Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Words** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Data**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

SPI serial bus configuration menu (on page 247)

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.

Field or control	Description
Display	Enables or disables displaying marks from this search on the bus waveform.
Search Type	Set to **Bus**.
Source	Select the **SPMI** bus for which to search.
Mark On	Select the type of information for which to search.
Slave Address	Sets the slave address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **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**.

Field or control	Description
Master Address	Sets the master address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Master Read**, **Master Write**, or **DD Block Master Read**.
Register Address	Sets the register address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Master Read**, **Master Write**, **Register Read**, **Register Write**, **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, or **Ext. Register Write Long**.
Data	Sets the data value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **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**.
Data Bytes	Sets the number of data bytes for which to search. 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 a value. Available when **Mark On** is **Extended Register Read**, **Extended Register Write**, **Ext. Register Read Long**, or **Ext. Register Write Long**.
No Response	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. Available when **Mark On** is **Master Read**, **Register Read**, **Extended Register Read**, **Ext. Register Read Long**, **DD Block Master Read**, **DD Block Slave Read**, or **Transfer Bus Ownership**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

SPMI serial bus configuration menu (on page 249)

SpaceWire serial bus search configuration menu

Use the SpaceWire Search configuration menu to define conditions to search for and mark on a SpaceWire bus waveform. You can have multiple searches on the same bus.

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Search Type	Set to **Bus**.
Source	Select the SpaceWire bus that you want to search.
Mark On	Select the type of information for which to search.
Control Code Type	Sets the control code type (**Null** or **Time Code**) for which to search and mark on the waveform. Available only when **Mark On** is **Control Code**.
Control Character Type	Sets the control character type (**Flow Control Token, Escape Code, or End Of Packet)**. Available only when **Mark On** is **Control Character**.
Data Bytes	Sets the number of data bytes for which to search (one to five bytes). Use the **A** knob to change the value. Available when **Mark On** is **Data**.
Data	Sets the data pattern for which to search. The number of bits shown depends on the **Data Bytes** setting. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Data**.
Errors	Sets the error type (**Error End of Packet, Escape, or Parity**) for which to search and mark on the waveform. Available when **Mark On** is **Errors**.
Time Code	Sets the time code pattern for which to search. The number of bits searched is 6. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available only when **Mark On** is **Control Code** and **Control Code Type** is **Time Code**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings. SpaceWire is currently not supported as a bus trigger source.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria. SpaceWire is currently not supported as a bus trigger source.

See also

Bus Search configuration menus (on page 375)

SpaceWire serial bus configuration menu (on page 245)

SVID serial bus search configuration menu

Use the SVID Search configuration menu to define the conditions to search for and mark on an SVID bus waveform. You can have multiple searches on the same bus.

SVID search menu, fields and controls

Field or control	Description
Display	Enables or disables displaying search marks for this search.
Search Type	Set to **Bus**.
Source	Select the SVID bus that you want to search.
Mark On	Sets the type of information for which to search. If the selected search type requires more input, the menu update shows additional fields.
Start	A **Start** event occurs when the SDA line transitions from a High to Low state while the SCL line remains High.
Slave Address	Sets the slave address value for which to search. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. Available when **Mark On** is **Slave Address**.
Command Type	Sets the search to mark **Command** events. Available Command Types are **Get**, **Set**, and **Optional**. Available when **Mark On** is **Command**.
Command Response	Sets the command response. Available when **Mark On** is **Command**. Following are the available command responses when **Version** is set to **Non VR14** in SVID serial bus configuration menu. (on page 250) ▪ For **Get**: GetReg. ▪ For **Set**: SetVID Fast, SetVID Slow, SetVID Decay, SetPS, SetRegAddr, and SetRegData. ▪ For **Optional**: Test Mode and SetWP. Following are the available command responses when **Version** is set to **VR14** in SVID serial bus configuration menu. (on page 250) ▪ For **Get**: GetReg, GetRegVendor, GetRegTestCfg, GetRegPktRecent, GetRegPktAlert, GetRegPktBad, and GetRegVrEvent. ▪ For **Set**: SetVID Fast, SetVID Slow, SetVID Decay, SetPS, SetRegAddr, SetRegData, SetRegAddrVendor, SetRegDataVendor, SetRegAddrTestCfg, and SetRegDataTestCfg. ▪ For **Optional**: Test Mode and SetWP.
Mode	Sets the mode of the payload for which to search. Available modes are **Master**, **Slave**, or **Either**. Available when **Mark On** is **Payload**.

Field or control	Description
Data	Sets the data for which to search for different modes. Available when **Mark On** is **Payload**. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
Errors	Sets the search to mark events that meet the specified address for a Device or register location. The **Errors** sets the number of bits in the field. Tap the **Binary** 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 **Mark On** is **Errors**.
Error Type	Set the error type from the drop down. Available when **Mark On** is **Errors**. Available error types are Any, NACK, Reject, Mixed, and Parity.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

See also

Bus Search configuration menus (on page 375)

SVID serial bus configuration menu (on page 250)

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 that the fields and controls changes based on the selected USB **Speed** and **Packet View** fields from the USB serial bus configuration menu (on page 251).

Field or control	Description
Display	Enables or disables displaying search marks on this search.
Act on Event	Tap the **Act on Event** button to configure the actions the instrument must take when a search event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Search Type	Set to **Bus**.
Source	Select the USB bus that you want to perform search upon.

Field or control	Description
Mark On	Sets the type of information to search for and mark on the bus waveform acquisition. Available options are **Compliance Pattern**, **Control Character**, **Custom Pattern**, **Data Packet**, **End of Packet**, **Error**, **Handshake Packet**, **Header Fields**, **Ordered Set**, **Packet**, **Redundancy**, **Reset**, **Resume**, **RS-FEC Sync Bits**, **Special Packet**, **START_RS_FEC**, **Suspend**, **Sync**, **Sync Bits**, **Sync Header**, and **Token Packet**. The field and control vary based on the instrument, speed, packet view, and lane type selected in the USB bus configuration menu and USB search configuration menu.
Lane	Sets the lane to search for. When **Speed** is **USB 3.2 Gen 1 (5 Gbps)** and **Lanes** is **Dual** the available options are **USB 3.2 Gen 1x2**, **Lane 0**, and **Lane 1**. When **Speed** is **3.2 Gen 2 (10 Gbps)** and **Lanes** is **Dual** the available options are **USB 3.2 Gen 2x2**, **Lane 0**, and **Lane 1**.
Sync	Sets the synchronization type to search for. Available when **Speed** is **USB 1.0 (1.5 Mbps)**, **USB 1.1 (12 Mbps)**, or **USB 2.0 (480 Mbps)** and **Mark On** is set to **Sync**.
Ordered Set	Sets the ordered set to search for. This search setting is available when **Mark On** is set to **Ordered Set**. The field and control vary based on the instrument, speed, packet view, and lane type selected in the USB bus configuration menu and USB search configuration menu. The available options are **DPPABORT**, **DPPEND**, **DPPSTART**, **DPHSTART**, **HPSTART**, **LCSTART**, **LOW POWER**,**SDS**, **SKIP**, **SKP**, **SLOS1**, **SLOS2**,**SYNC**, **TSEQ**, **TS1**, **TS2**, and **TSNOS**.
Sync Header	Sets the sync header to search for. This search setting is available when **Mark On** is set to **Sync Header** and the fields are **Control (1100b)**, **Data (0011b)**, and **Invalid**. Available when the **Speed** is set to **USB 3.1 Gen 2** or **USB 3.2 Gen 2** and **Packet View** is **Off**.
Sync Bits	Sets the sync bits to search for. It is available when • In the bus configuration menu when **Speed** is set to **USB 4.0 Gen 2 (10 Gbps), Packet View** is **Off,** and **Use RS-FEC** box is unchecked and in the search configuration menu **Mark On** is set to **Sync Bits**. The available fields are **Ordered Set(10), Data(01),** and **Invalid**. • In the bus configuration menu when **Speed** is set to **USB 4.0 Gen 3 (20 Gbps), Packet View** is **Off,** and **Use RS-FEC** box is unchecked and in the search configuration menu **Mark On** is set to **Sync Bits**. The available fields are **Ordered Set(1010), Data(0101),** and **Invalid**.

Field or control	Description
Character	Sets the control character to search for. This search setting is available when **Mark On** is set to **Control Character**. When **Speed** is in **USB 3.0 (5 Gbps)**, **USB 3.1 Gen 1 (5 Gbps)**, or **USB 3.2 Gen 1 (5 Gbps)**, the available options are **COM (K28.5)**, **EDB (K28.3)**, **END (K29.7)**, **EPF (K23.7)**, **SDP (K28.2)**, **SHP (K27.7)**, **SKP (K28.1)**, **SLC (K30.7)**, **SUB (K28.4)**, and **Any**. When **Speed** is in **USB 3.1 Gen 2 (10 Gbps)** or **USB 3.2 Gen 2 (10 Gbps)**, the available options are **SKP (CCh)**, **EDB (69h)**, **END (65h)**, **EPF (36h)**, **SDP (96h)**, **SDS (E1h)SHP (9Ah)**, **SKP (CCh)**, **SKPEND (33h)**, **SLC (4Bh)**, and **Any.**
Compliance	Sets the compliance pattern sequence to search for. This search setting is available when **Packet View** is **Off** and **Mark On** is set to **Compliance Pattern**. When **Speed** is in **USB 3.0 (5 Gbps)**, **USB 3.1 Gen 1 (5 Gbps)**, or **USB 3.2 Gen 1 (5 Gbps)**, the available options are: ▪ **CP1**: Nyquist frequency (D10.2) ▪ **CP3**: COM pattern (K28.5) ▪ **CP4**: The low frequency periodic signaling pattern (LFPS) ▪ **CP5/6**: K28.7 with/without de-emphasis ▪ **CP7/8**: Repeating 50-250 1's and then 50-250 0's When **Speed** is in **USB 3.1 Gen 2 (10 Gbps)** or **USB 3.2 Gen 2 (10 Gbps)**, the available options are: ▪ **CP9**: Complains pattern starts with SYNC to signal the beginning of the pattern and rest the scrambler. Symbol Number 0-15: SYNC ordered set and 16-65535: Datasymbol 00h that is scrambled. ▪ **CP10**: (AAh) Nyquist pattern at 10 Gb/s. This is not 128b132b encoded. ▪ **CP11**: (CCh) Nyquist/2 at 10 Gb/s. This is not 128b132b encoded. ▪ **CP13**: With pre-shoot (no de-emphasis). Repeating 64 1's and then 64 0's at 10 Gb/s. This is not 128b132b encoded. ▪ **CP14**: With de-emphasis (no pre-shoot). Repeating 64 1's and then 64 0's at 10 Gb/s. This is not 128b132b encoded. ▪ **CP15**: With pre-shoot and de-emphasis. Repeating 64 1's and then 64 0's at 10 Gb/s. This is not 128b132b encoded.
Symbols	Sets the symbols (length of the pattern between 1 to 16) to search for. This search setting is available when **Packet View** is **Off** and **Mark On** is set to **Custom Pattern**. The custom setting is available only for **USB 3.0 (5 Gbps)**, **USB 3.1 Gen 1 (5 Gbps)**, **USB 3.2 Gen 1 (5 Gbps)**, **USB 3.1 Gen 2 (10 Gbps)**, or **USB 3.2 Gen 2 (10 Gbps)** speed signals.
Symbol Field	Sets the symbols field to search for.
Handshake Type	Sets the handshake packet type to search for. This search setting is available when **Mark On** is set to **Handshake Packet**. Available options are **Any (XX10)**, **ACK (0010)**, **NAK (1010)**, and **STALL (1110)**.

Field or control	Description
Packet Type	Sets the packet type to search for. This search setting is available when **Mark On** is set to **Special Packet** for the USB serial bus signal speed of 1.5 Mbps, 12 Mbps, and 480 Mbps of all variants and is set to **Packet** for the USB serial bus signal speed of 5 Gbps, 10 Gbps, and 20 Gbps of all variants. Available options are: ▪ For USB 1.0/1.1/2.0 speed signal: **Any (XX00)**, **PRE (1100)**, and **RESERVED (0000)** ▪ For USB 3.0/3.1 Gen 1/3.2 Gen 1/USB 3.1 Gen 2/3.2 Gen 2 speed signal: **LMP**, **TP**, **DP**, **ITP**, and **ANY** ▪ For USB 4.0 Gen 2/Gen 3 speed signal: **Control**, **LMP**, **Read Write CS**, **Time-Sync**, **ANY.**
Error	Sets the error (**Character** or **Disparity**) to search for in 5 Gbps variants. This setting is available when **Packet View** is **Off** and **Error Type** is set to **Error**.
Hub Address	Sets the Hub address to search for. Tap the **Binary**, **Hex**, or **Decimal** 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. This search setting is available when **Mark On** is set to **Special Packet** and **Packet Type** is all.
Port Address	Sets the port address to search for. Tap the **Binary**, **Hex**, or **Decimal** 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. This search setting is available when **Mark On** is set to **Special Packet** and **Packet Type** is all.
Error Type	Sets the error type to search for. This search setting is available when **Mark On** is set to **Error**. Available options are **CRC**, **PID Check Bits**, **Token CRC5**, **Data CRC16**, **Bit Stuffing**, **Error, HEC, ECC, CRC32,** and **RS-FEC.** The **PID Check Bits**, **Token CRC5**, **Data CRC16**, and **Bit Stuffing** options are available only for **USB 1.0 (1.5 Mbps)**, **USB 1.1 (12 Mbps)**, and **USB 2.0 (480 Mbps)** speed signals. The CRC option is available only for **USB 3.0 (5 Gbps)**, **USB 3.1 Gen 1 (5 Gbps)**, **USB 3.2 Gen 1 (5 Gbps)**, **USB 3.1 Gen 2 (10 Gbps)**, **USB 3.2 Gen 2 (10 Gbps), USB 4.0 Gen 2 (10 Gbps),** or **USB 4.0 Gen 3 (20 Gbps)** speed signals.
Address	Sets the token packet address pattern to search for. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is in **Token Packet** and **Token Type** is set to all except **SOF (0101)**.
Token Type	Sets the token packet type to search for. This search setting is available when **Mark On** is set to **Token Packet**. Available options are **Any (XX01)**, **SOF (0101)**, **OUT (0001)**, **IN (1001)**, and **SETUP (1101)**.

Field or control	Description
Mark When / **Mark When Address**	Sets the mark when address to search for. The setting **Mark When** is available when **Mark On** is set to **Data Packet**. The setting **Mark When Address** is available when **Mark On** is set to **Token Packet** and **Token Type** is set to all except **SOF (0101).** Available options are **=**, **≠**, **<**, **>**, **≤**, **≥**, **Inside Range**, and **Outside Range**. When set to **Inside Range** or **Outside Range**, fields are displayed to set a high and low boundary pattern for the specified search range. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values.
Address High	Sets the token packet address high pattern to search for. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is in **Token Packet**, **Token Type** is set to all except **SOF (0101)** and **Mark When Address** is set to **Inside Range**.
Endpoint	Sets the token packet endpoint pattern to search for. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is in **Token Packet** and **Token Type** is set to all except **SOF (0101)**.
Frame Number	Sets the frame number pattern to search for. Use in conjunction with the **Mark When** field to specify the exact search condition. Tap the **Binary**, **Hex**, or **Decimal** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is in **Token Packet** and **Token Type** is set to **SOF (0101)** only.
Data Packet Type	Sets the data packet type to search for. This search setting is available when **Mark On** is set to **Data Packet**. Available options are **Any (XX11)**, **DATA0 (0011)**, and **DATA1 (1011)**.
Data Bytes	Sets the number of data bytes to search for (one to sixteen bytes). Tap the field and use the **A** knob to change the value. This search setting is available when **Mark On** is set to **Data Packet**.
Byte Offset	Sets the byte offset (Don't Care or the number of bytes). Tap the field and use the **A** knob to change the value. This search setting is available when **Mark On** is set to **Data Packet**.

Field or control	Description
Data	Sets the data packet pattern to search for. The number of bits shown depends on the Data Bytes setting. Use in conjunction with the Mark When field to specify the exact search condition. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is in **Data Packet** and **Mark When** is set to All except **Inside Range** and **Outside Range**.
Data Low	Sets the low data packet pattern to search for. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is in **Data Packet** and **Mark When** is set to **Inside Range** or **Outside Range**.
Data High	Sets the high data packet pattern to search for. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is in **Data Packet** and **Mark When** is set to **Inside Range** or **Outside Range**.
Protocol Defined Field	Sets the protocol defined field to search for. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is set to **Header Fields**.
Supplemental ID	Sets the supplemental ID to search for. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is set to **Header Fields**.
HOP ID	Sets the HOP ID to search for. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is set to **Header Fields**.
Header Length	Sets the header length to search for. Tap the **Binary** or **Hex** field and use the **A** and **B** knobs to select and change the values. Or double-tap on the field and use the virtual keypad to enter values. This search setting is available when **Mark On** is set to **Header Fields**.
A, B knob controls	Use the **A** knob to select (highlight) the digit(s) to change. Use the **B** knob to change the value of the digit(s).

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

See also

Bus Search configuration menus (on page 375)

USB serial bus configuration menu (on page 251)

Cycle Search configuration menu

Use the Cycle Search configuration menu to define the conditions to search for and mark on an cycle waveform. You can have multiple searches on the same cycle search.

To open the Cycle Search menu:

1. Tap the **Add New... Search** button on the Results bar.
2. Set the **Search Type** to **Cycle**.

Field or control	Description
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Search Type	Set to **Cycle**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	Defines the waveform cycle for which to search. ▪ **< Limit**: A cycle is less than the specified time limit. ▪ **> Limit**: A cycle is greater than the specified time limit. ▪ **= Limit**: A cycle is equal to the specified time limit. ▪ **≠ Limit**: A cycle does not equal (is greater than or less than) the specified time limit. ▪ **Inside Range**: A cycle is in the specified time range. ▪ **Outside Range**: A cycle is outside of the specified time range. Default is < Limit.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Calculated As	Sets the calculated as **Period** or **Frequency**. Default is period.

Polarity	Sets the polarity of the cycle to detect positive and negative cycle. Default is positive.
Limit	Sets the limit for the cycle search to occur.
High Limit	Sets the high limit for the measurement value. This control is only present when **Mark When** is set to **Inside Range** or **Outside Range**.
Low Limit	Sets the low limit for the measurement value. This control is only present **Mark When** is set to **Inside Range** or **Outside Range**.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus (on page 375)

Cycle Search configuration menu

Edge Search configuration menu (on page 472)

Pulse Width Search configuration menu (on page 475)

Rise/Fall Time Search configuration menu (on page 476)

Runt Search configuration menu (on page 477)

Timeout Search configuration menu (on page 480)

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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Edge**.

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Slope	Sets the signal transition direction to detect (rising or falling).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus (on page 375)

Cycle Search configuration menu (on page 471)

Pulse Width Search configuration menu (on page 475)

Rise/Fall Time Search configuration menu (on page 476)

Runt Search configuration menu (on page 477)

Timeout Search configuration menu (on page 480)

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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Logic**.
Use Clock Edge?	Enables or disables finding logic patterns that occur on the specified clock edge. Yes places marks on the clock waveform wherever the logic pattern occurs. No places marks on the input signal waveform(s) wherever the logic pattern occurs.

Field or control	Description
Logic Pattern: Define Inputs	Opens the Logic Search-Define Inputs configuration menu where you define the logic state (High, Low, or Don't Care), and the signal threshold level that defines the logic state for each analog or digital signal. See Setup and Hold Search - Define Inputs configuration menu (on page 479).
Mark When	Defines the waveform logic event to mark, when Use Clock Edge is set to No. • **Goes True**: All conditions change to a true state. • **Goes False**: All conditions change to a false state. • **Is True > Limit**: Condition remains true longer than a specified time. • **Is True < Limit**: Condition remains true for less than a specified time. • **Is True = Limit**: Condition remains true for a specified time (within ± 5%). • **Is True ≠ Limit**: Condition does not remain true for a specified time (within ± 5%).
Clock Source	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform. Available when **Use Clock Edge** is **Yes**.
Clock Edge	Sets the polarity of the clock edge (rising or falling) for evaluating the other menu conditions. The Logic menu also lets you set the clock edge to either edge. Available when **Use Clock Edge** is **Yes**.
Clock Threshold	Sets the threshold level that the clock signal must pass through to be considered a valid transition. The clock threshold value is independent of the input signal threshold(s). Available when **Use Clock Edge** is **Yes**.
Define Logic	Sets the logic condition that must occur with all inputs. • **AND**: All conditions are true. • **OR**: Any condition is true. • **NAND**: One or more conditions are true. • **NOR**: No conditions are true.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus (on page 375)

Cycle Search configuration menu (on page 471)

Edge Search configuration menu (on page 472)

Pulse Width Search configuration menu (on page 475)

Rise/Fall Time Search configuration menu (on page 476)

Runt Search configuration menu (on page 477)

Timeout Search configuration menu (on page 480)

Logic Search - Define Inputs configuration menu

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

To open the Logic Search-Define Inputs configuration menu:

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

Logic Search - Define Inputs configuration menu fields and controls

Field or control	Description
Ch(x) (analog channels)	Use to select the signal sources logic condition on which to perform the logic search (**High**, **Low**, **Don't Care**).
Set All	Sets all signal sources to detect a logic **High**, **Low**, or **Don't Care** condition.

Pulse Width Search configuration menu

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

To 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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Pulse Width**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	• **< Limit**: A pulse width is less than the specified time limit. • **> Limit**: A pulse width is greater than the specified time limit. **Inside Range**: A pulse width is in the specified time range.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Time Limit	Sets the time period condition to be met.
High Time Limit	Sets the longest acceptable pulse width time period for the range condition. Available only when **Mark When** is **Inside Range** or **Outside Range**.

Field or control	Description
Low Time Limit	Sets the shortest acceptable pulse width time period for the range condition. Available only when **Mark When** is **Inside Range** or **Outside Range**.
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus (on page 375)

Cycle Search configuration menu (on page 471)

Edge Search configuration menu (on page 472)

Rise/Fall Time Search configuration menu (on page 476)

Runt Search configuration menu (on page 477)

Timeout Search configuration menu (on page 480)

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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Rise/Fall Time**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	• **< Limit**: A signal has a rise/fall time less than the specified time limit. • **> Limit**: A signal has a rise/fall time greater than the specified time limit.

Field or control	Description
Time Limit	Sets the time period condition to be met.
Slope	Sets the signal transition direction to detect (rising or falling).
Upper Threshold	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus (on page 375)

Cycle Search configuration menu (on page 471)

Edge Search configuration menu (on page 472)

Pulse Width Search configuration menu (on page 475)

Runt Search configuration menu (on page 477)

Timeout Search configuration menu (on page 480)

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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Runt**.

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	• **Occurs**: A runt signal event occurs. • **< Limit**: A runt signal event occurs that has a pulse width less than the specified time limit. • **> Limit**: A runt signal event occurs that has a pulse width greater than the specified time limit.
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Time Limit	Sets the time period condition to be met. Available only when **Mark When** = **< Limit**, **> Limit**, **= Limit**, or **!= Limit**.
Upper Threshold	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus (on page 375)

Cycle Search configuration menu (on page 471)

Edge Search configuration menu (on page 472)

Pulse Width Search configuration menu (on page 475)

Rise/Fall Time Search configuration menu (on page 476)

Timeout Search configuration menu (on page 480)

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 .
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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Setup & Hold**.
Clock Source	Sets the signal to use as the clock. The clock source can be an analog, digital, math, or reference waveform.
Clock Level	Sets the threshold level that the clock signal must pass through to be considered a valid transition. The clock threshold value is independent of the input signal threshold(s).
Clock Edge	Sets the polarity of the clock edge (rising or falling) for evaluating the other menu conditions. The Logic menu also lets you set the clock edge to either edge.
Data Sources	Sets the data signal source(s). All selected sources must meet the specified setup and hold times. See Setup and Hold Search - Define Inputs configuration menu (on page 479).
Setup Time	Sets the length of time that data signal should be stable and not change before a clock edge occurs.
Hold Time	Sets the length of time that data signal should be stable and not change after a clock edge occurs.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus (on page 375)

Cycle Search configuration menu (on page 471)

Edge Search configuration menu (on page 472)

Pulse Width Search configuration menu (on page 475)

Rise/Fall Time Search configuration menu (on page 476)

Runt Search configuration menu (on page 477)

Timeout Search configuration menu (on page 480)

Setup and Hold Search – Define Inputs configuration menu

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

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

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

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

Field or control	Description
Ch(x) (analog channels)	Use to add (**Include**) or exclude (**Don't Include**) the data signal(s) from available input channels and waveforms. For analog channels, use the threshold field to set the data signal level that must be exceeded for the signal transition to be true.
Set All	Use to **Include** or **Don't Include** all available channels and waveforms as data signals.

Timeout Search configuration menu

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

To 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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Timeout**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	• **Stays High**: The signal stays above the specified threshold level longer than the specified time. • **Stays Low**: The signal stays below the specified threshold level longer than the specified time. • **Either**: The signal stays above or below the specified threshold level longer than the specified time.
Threshold	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Time Limit	Sets the time period condition to be met.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus (on page 375)

Cycle Search configuration menu (on page 471)

Edge Search configuration menu (on page 472)

Pulse Width Search configuration menu (on page 475)

Rise/Fall Time Search configuration menu (on page 476)

Runt Search configuration menu (on page 477)

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
Display	Sets the display of the mark icons to either **On** or **Off**. If you have multiple searches defined, the control turns off just the marks for the selected search.
Stop Acquisition if Event Found	Stops input acquisition when the search event occurs. Default is not enabled.
Search Type	Set to **Window**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Mark When	• **Enters Window**: The signal outside a window enters the window defined by the upper and lower threshold settings. • **Exits Window**: The signal exits the window defined by the upper and lower threshold settings. • **Inside > Limit**: The signal remains inside the window longer then the specified time limit. • **Outside > Limit**: The signal remains outside the window longer then the specified time limit.
Upper Threshold	Sets the amplitude threshold value for the upper edge of the window.
Lower Threshold	Sets the amplitude threshold value for the lower edge of the window.
Time Limit	Sets the time period condition to be met. Available only when Mark When = Inside > Limit or Outside > Limit.
Threshold Crossing (Trigger When = Outside > Limit)	• **Upper**: A signal remains above the upper threshold level for longer than the specified time limit before crossing the upper threshold level to a lower level. • **Lower**: A signal remains below the lower threshold level for longer than the specified time limit before crossing the lower threshold level to a higher level. • **Either**: A signal remains outside (above or below) the two threshold levels for longer than the specified time limit before crossing either threshold level. • **None**: A signal remains outside the two specified threshold levels for longer than a specified time limit.

Field or control	Description
Threshold Crossing (Trigger When = Inside > Limit)	• **Upper**: A signal remains between two thresholds for longer than the specified time limit before crossing through the upper threshold. • **Lower**: A signal remains between two thresholds for longer than the specified time limit before crossing through the lower threshold. • **Either**: A signal remains between two thresholds for longer than the specified time limit before crossing through either the upper or lower threshold. • **None**: A signal remains between two threshold levels for longer than a specified time limit.
Copy Trigger Settings to Search	Sets the search criteria to match the current oscilloscope trigger settings.
Copy Search Settings to Trigger	Sets the current oscilloscope trigger settings to match the search criteria.

Other search types

Bus Search configuration menus (on page 375)

Cycle Search configuration menu (on page 471)

Edge Search configuration menu (on page 472)

Pulse Width Search configuration menu (on page 475)

Rise/Fall Time Search configuration menu (on page 476)

Runt Search configuration menu (on page 477)

Timeout Search configuration menu (on page 480)

SIM configuration menu

Use the SIM configuration menu to configure the blocks, models, and test points, and to plot and analyze the model.

To open the SIM configuration menu:

- Tap on **Add New SIM** to create a simulation.
- To open the SIM configuration menu, double-tap the SIM badge on the Settings bar.

Field or control	Description
Workflow	Sets the workflow for the current SIM from the drop-down menu. The available workflow are • Simple De-embed • Simple Embed • Advanced • Rx Equalization Default is Simple De-embed. The displayed fields and controls may change depending on workflow selections.
System Analysis	Toggles the system analysis 2-port or 4-port. Refer to Configuring the blocks (on page 127) for more information.

Field or control	Description
Input Source(s)	Toggles the input sources to **Single** or **Dual**. Available when **System Analysis** is **4-port**. Refer to Configuring the blocks (on page 127) for more information.
Source 1	Sets the source 1 from the drop down list. The available input sources are channels, math waveforms, reference waveforms, and None. None disables the input; useful when creating test point filter files on one instrument for use on another with a different supported sample rate.
Source 2	Sets the source 2 from the drop down list. Available when **System Analysis** is **4-port** and **Input Source(s)** is **Dual**. If **Source 1** is set to **None**, then by default **Source 2** is **None**.
Sample Rate	Displays the sample rate for the source signal waveform(s) when **Source 1** is set to any channel and it cannot be edited. Sets the sample rate when **Source 1** is set to **None**. If the source waveforms are channels or math, it displays the current sample rate of the oscilloscope. If the source waveforms are reference waveforms, then it displays the sample rate of the reference waveforms. If sample rate are different for **Source 1** and **Source 2**, it display **Mismatched Sample Rates**.
Rx Filter	Select the Rx filter checkbox to enable Rx filter in the Simulation model. Default it is enabled. Available when **Workflow** is **Rx Equalization** or **Advanced** and receiver block is added.
CTLE	Select the CTLE checkbox to enable CTLE in the Simulation model. Default it is enabled. Available when **Workflow** is **Rx Equalization** or **Advanced** and receiver block is added.
FFE	Select the FFE checkbox to enable FFE in the Simulation model. Default it is enabled. Available when Workflow is **Rx Equalization** or **Advanced** and receiver block is added.
DFE	Select the DFE checkbox to enable DFE in the Simulation model. Default it is enabled. Available when Workflow is **Rx Equalization or Advanced** and receiver block is added.
Clock Recovery	Select the Clock Recovery checkbox to enable clock recovery in the Simulation model. Available when **Workflow** is **Rx Equalization**. When either the DFE or FFE checkbox is selected, the clock recovery checkbox is automatically enabled by default.
Global BW Limit	Displays the current bandwidth. Tap Global BW Limit to change the bandwidth to Auto, Full, or Custom bandwidth limit. Set the custom bandwidth by setting up following fields in the Global BW Limit configuration menu. Available when Workflow is Simple De-embed, Simple Embed, or Advanced.

Field or control	Description
Label	Double-tap the field to add a label to the SIM using the virtual keypad.
Signal Access Point	Sets the signal access point as End or Mid-circuit. It defines the point in the circuit where the oscilloscope captures the signal. End: The oscilloscope is positioned at the end of the circuit, acting as a receiver or load. Selecting End enables the Input Source selection (Single or Dual). Mid-Circuit: The oscilloscope is probing the signal somewhere within the transmission path.
Add Block	Adds the new blocks to the Physical/Simulation models. To add a block to the Physical/Simulation model: Tap the **Add Block** button. A new block labeled **New Block** appears in the Physical model. Select the block and drag to the desired location within the Physical/Simulation model and release to place it. Double-tap the block to open its configuration menu. Available when **Workflow** is **Simple De-embed**, **Simple Embed**, or **Advanced**. Refer to Configuring the blocks (on page 127) for more information. To delete, right-click the block and select delete or press Delete key on virtual keypad or keyboard.
Add Test Point	Adds new test points to the Physical or Simulation Model. Tap the **Add Test Point** button. A new test point labeled **New TP** appears in the Physical Model with an associated Math waveform in the Waveform View. Select the test point and drag to the desired location within the Physical/Simulation Model and release to place it. Double-tap the test point to open its configuration menu. Refer to Configuring the blocks (on page 127) for more information. To delete, right-click the test point and select delete or press Delete key on virtual keypad or keyboard.
Physical Model	The Physical Model is built by adding components (called blocks) and test points, with a maximum of eight user-defined blocks. It represents real-world in-circuit measurement elements such as probes, cables, and fixtures that can be de-embedded. In Advanced and Simple De-Embed workflows, all Physical Model blocks are de-embedded to calculate the Thevenin Equivalent Voltage (TEV). In the Simple Embed workflow, the acquired waveforms are used directly to calculate the TEV.
Simulation Model	The TEV represents the signal source for the Simulation Model, capturing the waveform either after de-embedding or directly from the acquired data. The Simulation Model uses this TEV as its input to run “what-if” analysis — such as embedding an alternate circuit, removing measurement path effects, or applying both changes to see their combined impact on the waveform.

Field or control	Description
Calculate SIM	Tap to calculate the SIM (Physical and Simulation Model).
Plot Test Points	Opens the test point plots for analyzing magnitude, phase, impulse, and step response of the system at that test point location. This button is enabled after Calculate SIM button is pressed and processing of the simulation is complete. Refer to Plots (on page 133) for more information. Double-tap the plot to maximize the plot view.
Receiver	Adds a Receiver block to the Simulation model. It opens the Receiver configuration window, where you can configure receiver settings and optimization options. Refer to Receiver configuration menu for more information.
Receiver Summary	Click **Receiver Summary** to view the configured receiver parameters and settings.
Set to Defaults	Click **Set to Defaults** to restore the defaults Rx Equalization settings.
OK	Tap to close the menu.

Block configuration menu

Use the Block configuration menu to configure the block in the model.

Field or control	Description
Model	Select the model from the drop-down list. Available models are ▪ Thru ▪ S-parameters ▪ RLC Network ▪ Transmission Line ▪ Transfer Function ▪ FIR ▪ Shunt ▪ High-Z probe The default selected model is Thru. If Model is S-parameters, Transfer function, FIR, or Shunt, load a valid S-parameters configuration file and tap anywhere outside a configuration menu to close the menu and enable the changes. To close the menu, set the **Block** to **Off** or select the **Model** as **Thru** and tap outside the menu.

Field or control	Description
Label	Tap the field to enter a model label using keyboard or double-tap to add a label to the model using the virtual keypad.
Block	Enables (**On**) or disables (**Off**) the block.
Type	Select the type of model from drop-down list depending on selected model. Available when **Model** is **S-parameters** or **RLC Network**. Available types for S-parameters are: ▪ 2-port ▪ 4-port Single-Ended ▪ 4-port differential ▪ 6-port ▪ 8-port ▪ 12-port ▪ 16-port Available types for RLC Network are: ▪ Series 1 ▪ Series 2 ▪ Shunt 1 ▪ Shunt 2 ▪ CRLC ▪ RLCC
Length Loss Scale Factor	Tap to enter a length loss scale factor using keyboard or virtual keypad. Available when **Model** is **S-parameters**, **Transfer Function**, or **FIR**.
Block Port x	Sets the port number for the analysis. When any port is changed from its drop-down menu, opposite port get adjusted automatically. Available when **Model** is **S-parameters**, **Transfer Function**, **FIR**, or **Shunt**. For S-parameters: Select the port number from **Block Port 1** to **Block Port 4** when **Type** is **2-port** to **16-port** analysis. For **Transfer Function** and **FIR** model, Block Ports are default and cannot be edited. For Shunt model, Block Port cannot be edited when the **Number of Port** is **1**. Set the Block Port 1 and Block Port 3 when the **Model** is **Shunt** and **Number of Port** is **2**.
Browse	Tap to navigate and select the S-parameter file. Available when **Model** is **S-parameters**, **Transfer Function**, **High-Z Probe**, **FIR** (or filter file in case of FIR Block), and **Shunt**.

Field or control	Description
Plot	Tap to plot all parameters. Plots the S-parameters, impedance, and Smith Charts of the block for quickly validating component-level behavior.
Permutation	Select the permutation as **Typical**, **VNA**, or **Alternate**. Available when **Model** is **S-parameters** and **Type** is **4-port differential**.
Frequency Spacing	Sets the frequency spacing interval for the parameters. The minimum frequency value is 1 MHz and maximum frequency value is 1 GHz. Available when **Model** is **RLC Network**, **Transmission Line**.
Resistance	Sets the resistance value. Available when **Model** is **RLC Network**.
Inductance	Sets the inductance value. Available when **Model** is **RLC Network**.
Capacitance	Sets the capacitance value. Available when **Model** is **RLC Network**.
C2	Sets the capacitance value. Available when **Model** is **RLC Network** and **Type** is **RLCC** or **CRLC**.
Z0	Sets the Z0 value. Available when **Model** is **Transmission Line**.
Delay	Sets the delay. Available when **Model** is **Transmission Line**.
Number of Ports	Toggles between the ports **1** or **2** for shunt model. Default is 1. Available when **Model** is **Shunt**.
Info	Tap for more information on selected model.

To delete, right-click the block and select delete or press Delete key on virtual keypad or keyboard.

Test Point configuration menu

Use the Test Point configuration menu to configure the test point in the model.

Field or control	Description
Label	Tap the label field to enter a test point label using keyboard or virtual keypad.
Global Delay	Toggles between **Remove** or **Keep**. Default is Remove.
Save Filter	Tap to save the filer. It opens a Save As dialog to save the file. See Test Point Save Filter configuration menu (on page 489) for more information.
Waveforms	Select the waveforms checkboxes. Available waveforms are Differential, Common Mode, Single Default is checked for Differential (D). At least one Waveforms checkbox must be selected.

Test Point Save Filter configuration menu

Use the Test Point Save Filter configuration menu to save filter files for use elsewhere. Tap on **Save Filter** from the Test Point configuration menu to save the filter files.

To save filter, the file is saved based on the System Analysis and Input Source. If **System Analysis** is **4-port** and **Input Source** is **Dual**, it saves 8 output filter files.

If **System Analysis** is **2-port** and **Input Source** is **Single**, it saves 1 output filter file only.

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Tap to open the **Browse Save As Location** configuration menu, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu (on page 318).
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save As Type	Default save SIM filter as .zip file.
Cancel	Cancels the file save action and closes the configuration menu.
Save	Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.

DUT SOURCE configuration menu

Use the DUT Source configuration menu to configure the DUT Source in the Advanced workflow.

1. Tap the **Add New SIM** or double-tap the SIM badge from the Settings bar to open the SIM configuration menu.
2. Tap **Workflow** and select **Advanced** from the drop-down list.
3. Double-tap the **DUT Source** from the Physical or Simulation Model.

Field or control	Description
Model	Select the model as **Nominal**, **s1p File**, or **s2p File**. The default model is Nominal. Once you browse and load any required file(s), tap anywhere outside the configuration menu to close it and enable the changes. If Model is selected as s1p File or s2p File, load a valid S-parameters configuration file, else select Nominal to close the menu.
Label	Tap the field to enter a label using keyboard or virtual keypad.
Browse	Tap to navigate and select the S-parameter file. Available when **Model** is **s1p File** or **s2p File** and **Tx Equalization** is **FIR Filter File** for simulation model only in Advanced workflow.
Plot	Tap plot to all S-parameters.
Impedance	Sets the impedance value. Available when **Model** is **Nominal**.
Port	Select the port from drop-down list. Available when **Model** is **s2p File**.
Advanced	Tap for more information on selected model. Available when **Model** is **s1p File** or **s2p File**.
Use the same source settings as the Physical Model	This is default setting to use the same DUT Source settings as the physical model. It is available for Simulation Model only when **Workflow** is **Advanced**.
Tx Equalization	Sets the equalization type as **None**, **Pre-Emphasis, De-Emphasis, 3-tap FFE**, and **FIR Filter File**. The default is None. Available for simulation model only.
Action	Select **Add** or **Remove** equalizer type. Available when **Tx Equalization** is **Pre-Emphasis, De-Emphasis,** or **3-tap FFE** for simulation model only in Advanced workflow.
Symbol Rate	Sets the symbol rate for the equalizer. Available when **Tx Equalization** is **Pre-Emphasis, De-Emphasis,** or **3-tap FFE** for simulation model only in Advanced workflow.
Pre-Emphasis	c Available when **Tx Equalization** is **Pre-Emphasis** for simulation model only in Advanced workflow.
De-Emphasis	Sets the De-Emphasis value. Available when **Tx Equalization** is **De-Emphasis** for simulation model only in Advanced workflow.
Pre (C_{-1})	Sets the pre-cursor value. Available when **Tx Equalization** is **3-tap FFE** for simulation model only in Advanced workflow.
Main (C_0)	Sets the main cursor value. Available when **Tx Equalization** is **3-tap FFE** for simulation model only in Advanced workflow.
Post (C_{+1})	Sets the post-cursor value. Available when **Tx Equalization** is **3-tap FFE** for simulation model only in Advanced workflow.

Field or control	Description
Plot	Tap to plot magnitude, phase, impulse response, and step response. Available when **Tx Equalization** is **Pre-Emphasis** or **De-Emphasis** for simulation model only in Advanced workflow.

File Viewer configuration menu

To open File Viewer configuration menu:

1. Tap on any added model **Block**, **DUT Source**, **Scope**, **SAP**, **Load**, or **Probe** from Physical or Simulation Model.
2. Configure the block **Model** other than **Thru**, **Nominal**, or **Auto** depending on available block menu.
3. Tap on **Browse**, navigate and select the S-parameters file. Tap **Open**.
4. Tap on imported **File** hyperlink. It displays file content in read only view. Use scroll or swipe to view the content.

Note that this menu is available in non Windows oscilloscopes. For Windows oscilloscope, the file opens in Notepad.

Global Bandwidth Limit configuration menu

Use this menu to set up how the global bandwidth limit filter will be applied to all test point waveforms. Under the Global Bandwidth Limit menu, three options are available, including the option to create a custom filter.

Field or control	Description
Global Bandwidth Limit	Displays the current bandwidth. Tap Global BW Limit to change the bandwidth to **Auto**, **Full**, or **Custom** bandwidth limit. ▪ **Auto**: All test point transfer functions will be checked. The lowest frequency having gain of +14 dB from the DC gain will be determined. The bandwidth limit filter will be set to the cutoff frequency value and it is used as the cutoff frequency for the bandwidth limit filter. ▪ **Full** : Sets the limit to half the model's specified sample rate (Nyquist rate). ▪ **Custom**: Allows you to create a bandwidth limit filter. The Custom option is most useful when the Auto bandwidth filter is not appropriate for your input data, or your test has specific bandwidth requirements. Refer to the Create a custom bandwidth limit filter (on page 132) for more information.
Bandwidth	Set the bandwidth of the source signal.
Stopband	Sets the stopband for the signal.
Attenuation	Sets the attenuation of the signal.
Save Filter	Saves the global bandwidth limit filter as a .flt file. See Global Bandwidth Limit Save Filter configuration menu for more information.

General tips

For NRZ signals, set the bandwidth 3 to 5 times the Nyquist frequency of the data rate to capture harmonics.

For PAM4 signals, a bandwidth of approximately 2 to 3 times the Nyquist frequency is often sufficient.

- If the bandwidth is set too low, important harmonics will be lost, distorting the waveform.
- If the bandwidth is set too high, you may retain unnecessary noise and ripple from the model's upper frequency range.
- When in doubt, start with the Auto setting. Then, check the eye diagrams and jitter results. If they degrade when you lower the bandwidth, it means the bandwidth set is too low.

Global Bandwidth Limit Save Filter configuration menu

Tap **Save Filter** in the Global Bandwidth Limit Save Filter configuration menu to save the Global Bandwidth Limit filter as a single .flt output file.

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Tap to open the **Browse Save As Location** configuration menu, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu (on page 318).
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save As Type	Default save SIM filter as .flt file.
Cancel	Cancels the file save action and closes the configuration menu.
Save	Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.

Advanced configuration menu

To open Advanced configuration menu,

1. Tap on any added model **Block**, **DUT Source**, **Scope**, **SAP**, **Load**, or **Probe** from Physical or Simulation Model.
2. Configure the block **Model** other than **Thru**, **Nominal**, or **Auto** depending on available block menu.
3. Tap on **Browse**, navigate and select the S-parameters file. Tap **Open**.
4. Tap on **Advanced** button.

   If there are two Browse fields, depending on the configuration made in any block, then there will be two tabs corresponding to each file. The tabs and menus changes based on imported S-parameters files. It display the errors, warnings, and adjustment made in the S-parameters.

- Passivity
- Port Assignments
- Reference Impedance
- Uniform Frequency Spacing
- Extrapolate to DC

Tap **Plot** to plot the S-parameters Before and After Plots.

These plots allows you to compare it original S-parameters against the converted S-parameters that SIM generate when you change the reference impedance, change the frequency spacing, or extend the frequency range down to DC.

Field or control	Description
Plot	Tap to plot the all parameters.
Frequency Spacing	Displays the value of frequency spacing.
Save	Saves adjusted S-parameters information. Click **Save** to open the **Save As** dialog box and save the file. For more information, see Advanced Save configuration Menu.

Advanced Save configuration Menu

Use the Advanced Save Configuration menu to save filter files for use elsewhere. Tap on **Save** from the Advanced configuration menu to save the filter files.

To save filter, the file is saved based on the System Analysis and Input Source. If **System Analysis** is **4-port** and **Input Source** is **Dual**, it saves 8 output filter files.

If **System Analysis** is **2-port** and **Input Source** is **Single**, it saves 1 output filter file only.

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Tap to open the **Browse Save As Location** configuration menu, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu (on page 318).
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.

Field or control	Description
Save As Type	Default save SIM filter as .s4p file.
Cancel	Cancels the file save action and closes the configuration menu.
Save	Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.

Load configuration menu

Use the Load configuration menu to configure the Load in the Advanced workflow.

To configure the Load in the simulation Model:

1. Tap the **Add New SIM** or double-tap the SIM badge from the Settings bar to open the SIM configuration menu.
2. Tap **Workflow** and select **Advanced** from the drop-down list.
3. Double-tap the **Load** from the Simulation Model.

Field or control	Description
Model	Select the model as **Nominal**, **s1p File**, or **s2p File**. The default model is Nominal. If Model is selected as s1p File or s2p File, load a valid S-parameters configuration file, else select Nominal to close the menu.
Label	Tap the field to enter a label using keyboard or virtual keypad.
Plot	Tap to plot the all parameters.
Impedance	Sets the impedance value. Available when **Model** is **Nominal**.
Browse	Tap to navigate and select the S-parameter file. Available when **Model** is **s1p File**,or **s2p File**.
Port	Select the port from drop-down list. Available when **Model** is **s2p File**.
Info	Tap for more information on selected model. Available when **Model** is **s1p File**,or **s2p File**.

Probe configuration menu

Use the Probe configuration menu to configure the Probe in the Advanced workflow.

To configure the Probe in the Physical model:

1. Tap the **Add New SIM** or double-tap the SIM badge from the Settings bar to open the SIM configuration menu.
2. Tap **Workflow** and select **Advanced** from the drop-down list.
3. Double-tap the Probe from the Physical model.

Field or control	Description
Model	Toggles between **Auto**, **Ideal** and **Custom**. Default is Auto.

Field or control	Description
Probe readout	▪ Probe readout provide information about connected probe, adapter, and tip are attached to the specified model source input. ▪ If there is no probe attached, then the message says: **No probe attached, using Ideal Probe instead.** ▪ If there is a probe attached but there are no s-parameters available, then the message says: **No s-parameters available for attached probe, using Ideal Probe instead.**
Label	Tap the field to enter a label using keyboard or virtual keypad. Default is Probe.
Plot	Tap to plot the all parameters.
Browse	Tap to navigate and select the S-parameter file. Available when **Model** is **Custom**.
Port	Select the port from drop-down list.
Info	Tap for more information on selected model.

SAP configuration menu

Use the SAP configuration menu to configure the SAP in the Advanced workflow.

To configure the SAP in the Physical model:

1. Tap the **Add New SIM** or double-tap the SIM badge from the Settings bar to open the SIM configuration menu.
2. Tap **Workflow** and select **Advanced** from the drop-down list.
3. Double-tap the SAP from the Physical model.

Field or control	Description
Model	Toggles between **Thru**, **s3p** and **s6p**. The default model is Thru. If Model is selected as s3p or s6p, load a valid S-parameters configuration file. To close the menu, set the **Block** to **Off** or select the **Model** as **Thru** and tap outside.
Block	Enables (**On**) or disables (**Off**) the block. Available when **Model** is **s3p** or **s6p**.
Label	Tap the field to enter a label using keyboard or virtual keypad. Default is SAP.
Block Port x	Select the port from drop-down list. Available when **Model** is **s3p** or **s6p**.
Browse	Tap to navigate and select the S-parameter file. Available when **Model** is **s3p** or **s6p**.
Plot	Tap to plot the all parameters.
Info	Tap for more information on selected model. Available when **Model** is **s3p** or **s6p**.

S-parameter Plot configuration menu

Double-tap on the **Plot** button wherever it is available in any menu to maximize the plot view.

Field or control	Description
View	Display the plots in single-ended or differential mode as selected. Available only when **Type** is **4-port differential** in the Block configuration menu.
Overlay	Select the overlay from drop-down list. Available options are: **Off**: Displays only the S-parameter(s) you select. Default is Off. **Transmission**: Displays forward path gain or loss (insertion loss) between ports to evaluate the channel attenuation. Examples: S21, S12, S43, S34 **Reflection**: Displays signal reflections (return loss) at the ports to assess impedance mismatch. Examples: S11, S22, S33, S44 **Crosstalk**: Displays the cross coupling between ports, including near-end (NEXT) and far-end (FEXT) interference. Examples: ▪ NEXT: S31, S13, S42, S24; ▪ FEXT: S32, S23, S41, S14 **Custom**: Allows you to select and overlay any combination of the S-parameters for comparison.
Select Parameters	Select the parameters boxes based on the plot requirement. Default, all parameters boxes are checked. If any of the parameters are unchecked, **Select All** button is enabled to select all parameters at once. If all parameters are selected, **Deselect All** button is enabled to deselect all parameters at once.
Plot Type	Select a plot type from drop-down list to determine what parameter is plotted in the S-parameter plots. Available plot types are **Magnitude**, **Phase**, **Impulse**, and **Step**. Default is Magnitude. Available when **Overlay** is **Off**. Double-tap the plot to maximize the plot view.
Impedance Plots	Select the checkbox to overlay all the impedance plots. Default is unchecked.
Smith Charts	Select the checkbox to overlay all the smith charts. Default is unchecked.
Frequency Range	Sets the frequency range as **Full** or **Limited**. Default, it is Full.
High Frequency	Sets the maximum frequency that plot is based on. Available when **Frequency Range** is **Limited**.
Low Frequency	Sets the minimum frequency that plot is based on. Available when **Frequency Range** is **Limited**.

Field or control	Description
Save Image	Tap to open the Save As menu. Navigate to the location where you want to save the plot image. Enter a file name in the File Name field. Select the image file format (PNG, BMP, or JPG). Tap save the plot image or tap cancel to close the menu.
Done	Tap to close the overall plot view.

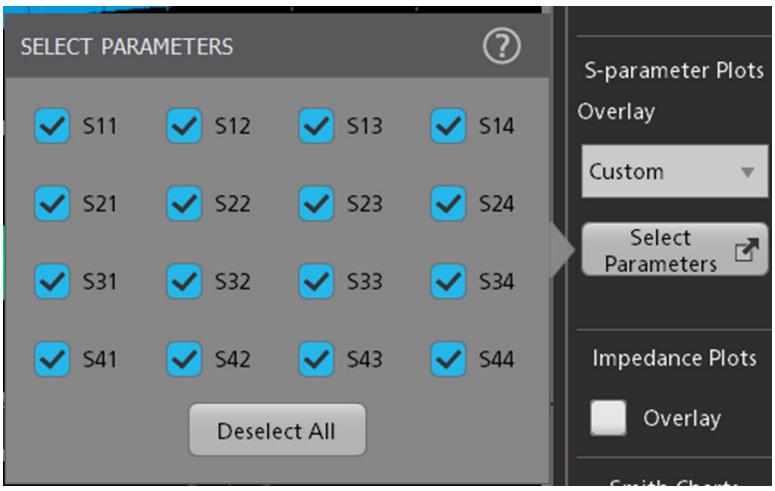
Plot image Save as menu

Field or control	Description
Save Location	Sets the location of the save file. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the 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.
Browse	Tap to open the Browse Save As Location dialog, to navigate to and select the location to save the file to.
File Name	Sets the file name of the file. The default file name is Tek000. The file name increments on the next save (Tek001, Tek002, and so on). Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save As Type	Select the image file format (PNG, BMP, or JPG).
Save	Tap save the plot image.
Cancel	Tap to close the menu.

Select Parameters configuration menu

To open Select Parameter configuration menu:

1. Tap on Block.
2. Select required **Model**.
3. Tap on **Plot** button. Plot is available for all the models other than Thru.
4. Select **S-parameters Plots Overlay** to **Custom**.
5. Tap **Select Parameters**.
6. Select the parameters checkboxes based on the plot requirement. Default, all parameters boxes are checked.



Default, all parameters are selected, **Deselect All** button is enabled to deselect all parameters at once.

If any of the parameters are unchecked, **Select All** button is enabled to select all parameters at once.

Scope configuration menu

Use the Scope configuration menu to configure the oscilloscope in the Advanced workflow.

To configure the Scope in the Physical Model:

1. Tap the **Add New SIM** or double-tap the SIM badge from the Settings bar to open the SIM configuration menu.
2. Tap **Workflow** and select **Advanced** from the drop-down list.
3. Double-tap the **Scope** from the Physical Model.

Field or control	Description
Model	Select the model as **Auto**, **Nominal**, **s1p File**, or **s2p File**.
Label	Tap the field to enter a label using keyboard or virtual keypad.
Plot	Tap to plot the all parameters.
Impedance	Sets the impedance value. Available when **Model** is **Nominal**.
Browse	Tap to navigate and select the S-parameter file. Available when **Model** is **s1p File**,or **s2p File**.
Port	Select the port from drop-down list. Available when **Model** is **s2p File**.
Info	Tap for more information on selected model. Available when **Model** is **s1p File**,or **s2p File**.

Receiver configuration menu

To open the Receiver configuration menu:

- Add a Receiver block to the simulation model.
- It opens the Receiver configuration window.

Field or control	Description
Setting	
Receiver	Enables or disables the receiver.
Receiver Type	Select the receiver type. Available options are Custom and Standard.
Standard	Selects the standard. If the selected first-level standard has multiple sub-standard variants, a slideout displays the available variants. Available when **Receiver Type** is **Standard**.
Receiver summary	Displays the operating parameters for each receiver block in a concise view. Available when **Receiver Type** is **Standard**.
Set to Defaults	Resets the receiver definition to the state in which it was first selected from the Standard drop-down list. Available when **Receiver Type** is **Standard**.
Label	Specifies the lable for the receiver block.
Rx Filter	Select the Rx filter checkbox to enable Rx filter in the Receiver block. Default it is enabled. Available when **Workflow** is **Advanced**.
CTLE	Select the CTLE checkbox to enable CTLE in the Receiver block. Default it is enabled. Available when **Workflow** is **Advanced**.
FFE	Select the FFE checkbox to enable FFE in the Receiver block. Default it is enabled. Available when Workflow is **Advanced**.
DFE	Select the DFE checkbox to enable DFE in the Receiver block. Default it is enabled. Available when Workflow is **Advanced**.
Clock Recovery	Select the Clock Recovery checkbox to enable clock recovery in the Receiver blockl. Available when **Workflow** is **Advanced**. When either the DFE or FFE checkbox is selected, the clock recovery checkbox is automatically enabled by default.
Optimization	Displays the optimization panel when one of the following conditions is true. Otherwise, the panel is not shown. ▪ A non-DDR standard-based receiver is in use. Optimization will select the optimal CTLE curve. ▪ A DDR receiver is in use. Optimization will select optimal DFE Gain. ▪ A Custom receiver is in use and a .JSON file containing multiple CTLE curves has been loaded into the CTLE block.
Eye Height	Select **Eye Height** to optimize the receiver based on the minimum vertical eye opening at the center of the recovered unit interval.
Eye Width	Select **Eye Width** to optimize the receiver based on the minimum horizontal eye opening at the center of the recovered unit interval. Available when DFE is enabled and tap adaptation is set to PK-PK.

Field or control	Description
Eye Area	Select **Eye Area** to optimize the receiver based on the eye opening area. Available when DFE is enabled and tap adaptation is set to PK-PK.
Optimize Receiver	Click **Optimize Receiver** to run the receiver optimization using the selected optimization metric.
Save Results	Click Save Results to save the optimization results. For more information, see Receiver Optimization Save Results configuration menu.

Receiver Optimization Save Results configuration menu

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Tap to open the **Browse Save As Location** configuration menu, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu (on page 318).
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek006. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save As Type	Default save optimization results as .csv file.
Cancel	Cancels the file save action and closes the configuration menu.
Save	Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.

Receiver Parameters configuration menu

Field or control	Description
Receiver Parameters	Displays the configured receiver parameters, including Rx filter, CTLE, FFE, DFE, and Clock Recover setting.

Test point Math waveform configuration menu

Use this menu to set math waveform parameters. To access the Math menu, double-tap a Math waveform badge.

Field or control	Description
Display	Enables (**On**) or disables (**Off**) displaying the math waveform in the Waveform View window.
Vertical Scale	Sets the vertical scale using the multipurpose knob, double-tap to bring up the virtual keypad, or tap the up and down arrows to change the scale.
Auto Scale	Select the Auto Scale checkbox to calculates the vertical scale and position to center and display the entire waveform.
Label	Double-tap the field to add a label to the math waveform using the virtual keypad.
Position	Sets the vertical position of the math waveform.
Set to 0	Sets the vertical position of the math waveform to zero (vertical center of a slice (Stacked mode) or the screen (Overlay mode).

Clock Recovery configuration menu

Use the Clock Recovery configuration menu to configure the Clock Recovery in the Rx Equalization workflow.

1. Tap the **Add New SIM** or double-tap the SIM badge from the Settings bar to open the SIM configuration menu.
2. Tap **Workflow** and select **Rx Equalization** from the drop-down list.
3. Double-tap **Clock Recovery** from the Simulation Model.

Field or control	Description
Method	Sets the method as **PLL** or **Explicit Clock**. By default it is PLL.
Symbol Rate Detection	Sets the symbol rate as **Auto** or **manual**. By default it is Auto. Available when **Method** is **PLL**.
Nominal Symbol Rate	Sets the nominal symbol rate. Available when **Method** is **PLL** and **Symbol Rate Detection** is **Manual**.
PLL Method	Sets the PLL method as **Type I** or **Type II**. Available when **Method** is **PLL**.
Loop BW	Sets the loop bandwidth of the PLL. Available when **Method** is **PLL**.
JTF BW	Sets the jitter transfer function bandwidth. Available when **Method** is **PLL**, and **PLL Method** is **Type II**.
Damping	Sets the damping value. Available when **Method** is **PLL**, and **PLL Method** is **Type II**.
Clock Source	Select the source channel from the drop-down list. Available when **Method** is **Explicit Clock**.
Clock Offset	Sets the clock offset as **Auto** or **Manual**. By default it is Auto. Available when **Method** is **Explicit Clock**.
Offset	Sets the offset value. Available when **Method** is **Explicit Clock** and **Clock Offset** is **Manual**.

RX Filter configuration menu

Use the Clock Recovery configuration menu to configure the Clock Recovery in the Rx Equalization workflow.

1. Tap the **Add New SIM** or double-tap the SIM badge from the Settings bar to open the SIM configuration menu.
2. Tap **Workflow** and select **Rx Equalization** from the drop-down list.
3. Double-tap **Rx Filter** from the Simulation Model.

Field or control	Description
Type	Displays the type of the filter.
Bandwidth	Sets the bandwidth of the filter.
Plot	Tap to plot the filter response.

CTLE configuration menu

Continuous Time Linear Equalizers (CTLE) are implemented to correct for channel losses and reflections at high frequencies.

Use the CTLE configuration menu to configure the CTLE in the Rx Equalization workflow.

1. Tap the **Add New SIM** or double-tap the SIM badge from the Settings bar to open the SIM configuration menu.
2. Tap **Workflow** and select **Rx Equalization** from the drop-down list.
3. Double-tap **CTLE** from the Simulation Model.

Field or control	Description
Type	Sets the type of the equalizer as **Custom, File** or **Standard**. Standard is available when receiver type is standard. It shows the CTLE presets based on the selected standard.
ADC	Sets the DC gain of the equalizer. Available when **Type** is **Custom**.
Zeros	Sets the number of zeros. Available when **Type** is **Custom**.
Poles	Sets the number of poles. Available when **Type** is **Custom**.
Zero frequencies or **Pole frequencies**	Sets the frequencies of Zeros/Poles. Available when **Type** is **Custom**.
CTLE File	Tap **Browse** and upload a JSON file with the CTLE presets to use. Available when **Type** is **File**.
Plot	Tap to plot the equalizer response.

FFE/DFE configuration menu

Use the FFE/DFE configuration menu to configure the FFE/DFE in the Rx Equalization workflow.

1. Tap the **Add New SIM** or double-tap the SIM badge from the Settings bar to open the SIM configuration menu.
2. Tap **Workflow** and select **Rx Equalization** from the drop-down list.
3. Double-tap **FFE/DFE** from the Simulation Model.

The fields and controls change based on the **FFE** and/or **DFE** checkbox(s) selected from the menu.

Settings tab

Field or control	Description
Tap Adaptation	Sets the tap adaptation as **MMSE, Pk-Pk, or None**. Default is MMSE. The MMSE option computes the tap values such that mean square error in minimal whereas None specifies the FFE/DFE tap values. Pk-Pk is available when only DFE box is selected.
Load From File	Tap Browse to select the file. Available when **Tap Adaptation is None.**
DFE Taps	Sets the number of DFE taps.
DFE Gain	Sets the gain to the input of DFE and Clock Recovery blocks.
Specify Tap Limits	Select the box to specify tap limits. Available when **Tap Adaptation is MMSE** or **Pk-Pk**.
Specify Tap Resolution	Select the box to specify tap resolution. Available when **Tap Adaptation is MMSE**.
Resolution	Sets the resolution value. Available when **Tap Adaptation is MMSE** and **Specify Tap Resolution** box is checked.
FFE Taps	Sets the number of FFE taps value.
FFE Normalization	Sets the normalization with the following choices **Unity Sum**, **L1**, **Main cursor**, and **None**.
Reference Tap	Sets the location of the reference tap, defining the main cursor in the FFE. The value is a 1 based index from 1 to the number of FFE Taps.
Taps per UI	Sets the number of taps per unit interval.
Save to File	Saves the FFE or DFE tap values to a JSON file. Click Save to File to open the Save As dialog box and save the file. For more information, see FFE/DFE Save to File configuration menu.

DFE tab

Tap DFE tab to display DFE Tap values.

Field or control	Description
Tap	The number of taps are available based on value set is **DFE Taps** field in the settings tab.
Lower Limit	Specifies the lower limit of the DFE tap value.
Tap Value	Displays the computed tap value when **Tap Adaptation** is **MMSE** or **Pk-Pk**. Enter the tap value when **Tap Adaptation** is **None**.
Upper Limit	Specifies the upper limit of the DFE tap value.

FFE Tap values

Tap FFE tab to display FEE Tap values.

Field or control	Description
Tap	The number of taps are available based on value set is **FFE Taps** field in the settings tab.

Field or control	Description
Tap Value	Displays the computed values when **Tap Adaptation** is **MMSE**. Enter the tap value when **Tap Adaptation** is **None.**

FFE/DFE Save to File configuration menu

Use the FFE/DFE Save to File configuration menu to save filter files for use elsewhere. Tap on **Save** from the FFE/DFE configuration menu to save the filter files.

To save filter, the file is saved based on the System Analysis and Input Source. If **System Analysis** is **4-port** and **Input Source** is **Dual**, it saves 8 output filter files.

If **System Analysis** is **2-port** and **Input Source** is **Single**, it saves 1 output filter file only.

Field or control	Description
Save Location	Lists the location where the file will be saved. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location. Or double-tap on the file name to open the virtual keyboard and enter a path. Tap the down arrow icon on the right end of the field to open a list of recent file save locations for the current save type.
Browse	Tap to open the **Browse Save As Location** configuration menu, to navigate to and select the location to which to save the file. See Browse Save As Location configuration menu (on page 318).
File Name	The file name assigned to the file. The default value is either the user-entered name used to last save this file type, or a numeric value calculated by the instrument if this file type has not previously been saved with a custom file name. The default value is Tek000. Tap the down arrow on the right edge of the field to display and select from a list of recently-saved file names. Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the file name to open the virtual keyboard and enter a file name.
Save As Type	Default save DFE/FFE taps as .json file.
Cancel	Cancels the file save action and closes the configuration menu.
Save	Saves the file to the specified location, closes the Save As configuration menu, and displays a confirmation message.

TDR measurement overview

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

To open the TDR measurement configuration menu for a measurement, double-tap a TDR measurement badge in the Results bar.

TDR Measurement configuration menu fields, controls, and panels

Field, control, or panel	Description
Configure panel	Sets the source, label text, and other fields that are specific to each measurement type. See Configure panel (TDR measurement configuration menu) (on page 505).
Reference Levels panel	Sets the reference levels and units used to take measurements, the scope of the reference level settings (global or local), and the method used to calculate the Top and Base waveform values. See Reference Levels panel (Measurement configuration menu) (on page 303).
Gating panel	Sets the measurement region (gate) used to take measurements. Select the scope of the gate setting (global or local), and the type of gating to use. See Gating panel (Measurement configuration menu) (on page 309).
Pass/Fail Testing panel	Sets the conditions for pass/fail testing for measurements and sets actions to take when a measurement fails. See Pass/Fail Testing panel (Measurement Settings menu) (on page 311).

Configure panel (TDR measurement configuration menu)

Use the configure panel to change the parameters of the measurement.

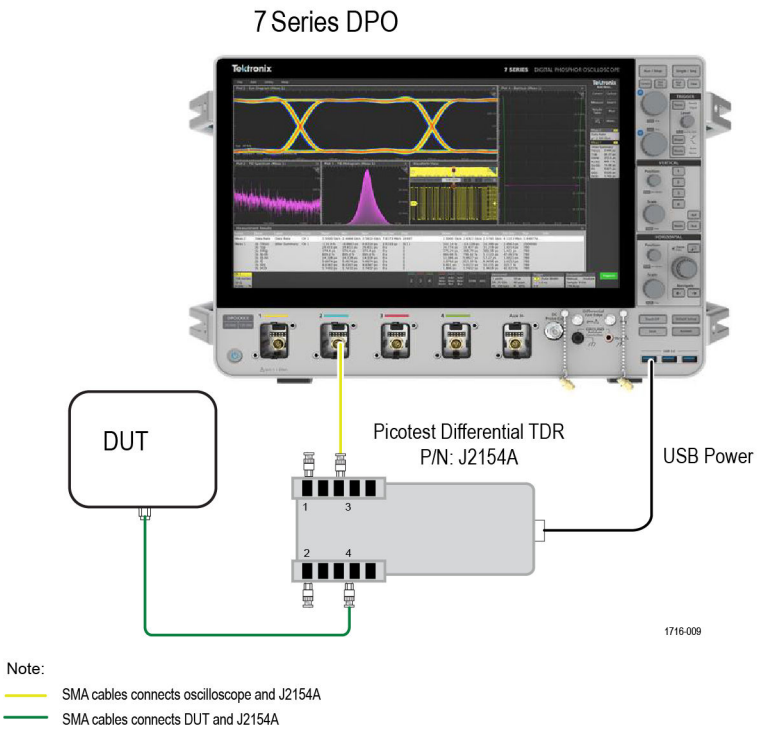
Double-tap a TDR measurement badge to open the TDR measurement configuration menu.

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

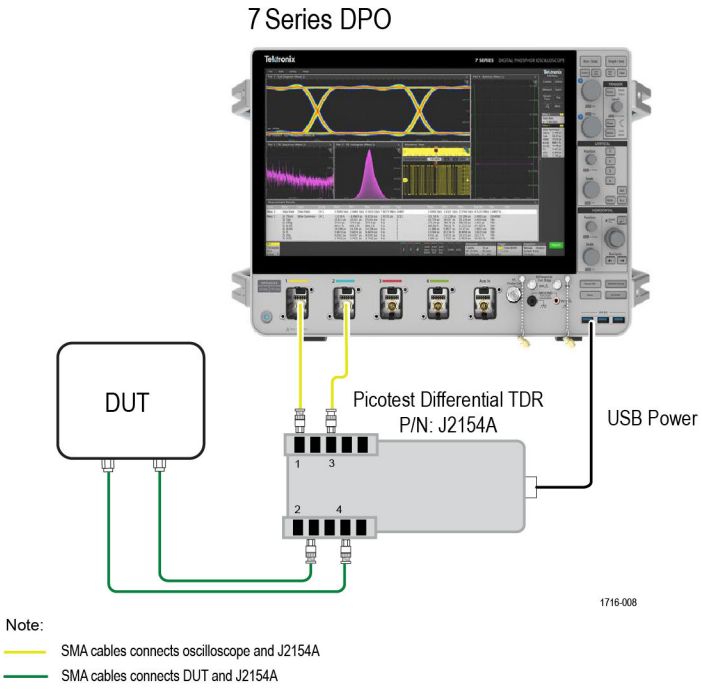
TDR Preset

The TDR Preset significantly relies on the specific TDR source used in the measurement system.

This feature sets the vertical settings of voltage sources and trigger source of the oscilloscope to the TDR source channel. TDR preset calibrates each line of TDR using average acquisition mode. It also sets the optimal sample rate and horizontal scale to accommodate incident and first reflection of the voltage waveform.



Connection diagram for TDR single-ended



Connection diagram for TDR differential

NOTE: The TDR Preset feature supports only the Picotest J2154A as a TDR source and the trigger is always set for negative TDR originating from the port 3.

Use the following steps to run a TDR Preset on the TDR measurements:

1. Add the TDR measurement.
2. Double-tap the TDR measurement results badge for the required measurement.
3. Configure the TDR measurement voltage sources based on the signal type.
4. Connect port 1 and port 3 of J2154A to the oscilloscope channels using identical SMA cables. Refer to the figure "Connection diagram for TDR differential" above.
5. For Single-Ended TDR, connect high-quality 50 Ω SMA cable to the port 4 of J2154A. The SMA cable should be terminated at far ends with exactly the same termination as the DUT termination. Refer the figure "Connection diagram for TDR single-ended" above.
6. For Differential TDR, connect high quality identical 50 ohm SMA cables to the port 2 and port 4 of J2154A. The SMA cable should be terminated at far ends with exactly the same termination as the DUT termination. Refer to the figure "Connection diagram for TDR differential" above.
7. Tap **TDR Preset** button for the autoset and calibration.

This performs the following settings:

- Sets the vertical channel settings for 50 Ω termination with DC coupling.
- Sets the horizontal settings for optimal sample rate and horizontal scale as per detected reflection.
- Performs calibration for each source and apply those calibration for the measurement instance.
- Refer the details on TDR preset algorithm (on page 610).

NOTE: In case of failure, the instrument displays a popup error message. See TDR badge errors and warning messages (on page 613).

The TDR Preset is specific to a previously calibrated termination. Rerun the TDR Preset whenever there is a significant change in the DUT termination.

Trigger configuration menu overview

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

To quickly change the trigger source from one channel to another when using a single source trigger type, single-tap on the **Trigger** badge. Select the new trigger source from the list.

To access the Trigger configuration menu, double-tap the **Trigger** badge on the Settings bar. The Trigger menu opens to show the settings for the current trigger type.

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

- Edge Trigger menu (on page 509)
- Pulse Width Trigger menu (on page 511)
- Rise/Fall Time Trigger menu (on page 513)
- Runt Trigger menu (on page 516)
- Sequence Trigger menu
- Timeout Trigger menu (on page 521)
- Window Trigger menu

Cycle Trigger configuration menu

Use the Cycle Trigger menu to trigger the instrument on specific cycle, including when a cycle is within or outside of specified limits.

To open the Cycle trigger menu:

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

Settings panel (Cycle Trigger configuration menu) - fields and controls

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Trigger When	Defines the waveform cycle on which to trigger. ▪ **< Limit**: A cycle is less than the specified time limit. ▪ **> Limit**: A cycle is greater than the specified time limit. ▪ **= Limit**: A cycle is equal to the specified time limit. ▪ **≠ Limit**: A cycle does not equal (is greater than or less than) the specified time limit. ▪ **Inside Range**: A cycle is in the specified time range. ▪ **Outside Range**: A cycle is outside of the specified time range. Default is < Limit.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Calculated As	Sets the calculated as **Period** or **Frequency**. Default is period.
Polarity	Sets the polarity of the cycle to detect positive and negative cycle. Default is positive.
Logic Qualification	Set to **On** to enable logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event. Default is off.
Define Inputs	Opens the Logic Qualification - Define Inputs dialog. Use this dialog to set the logic state, threshold levels, and logic operation of the input signals. Available when **Logic Qualification** is set to **On**.
High Limit	Sets the high limit for the measurement value. This control is only present when **Trigger When** is set to **Inside Range** or **Outside Range**.
Low Limit	Sets the low limit for the measurement value. This control is only present **Trigger When** is set to **Inside Range** or **Outside Range**.

Mode & Holdoff panel (Cycle Trigger configuration menu) - fields and controls

Field or control	Description
Trigger Mode	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event: **Auto** trigger mode enables the instrument to acquire and display a waveform even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **User Holdoff** sets the instrument to wait for a user to trigger the another trigger event. It is available when enabled through PI command. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Act on Trigger	Tap the **Act on Trigger** button to configure the actions the instrument must take when a trigger event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT.

Edge Trigger configuration menu

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

To open the Edge trigger menu:

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

Settings panel (Edge Trigger configuration menu) fields and controls

Field or control	Description
Trigger Type	Set to **Edge**.
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control. **AC Line** source (**Edge** trigger): 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.
Coupling	Set the conditioning to apply to the source signal trigger circuit from the source signal. **DC** coupling passes all input signals directly to the trigger circuitry. **Noise Reject** coupling provides stable triggering by increasing the trigger hysteresis. Increased hysteresis reduces the trigger sensitivity to noise so may require greater signal amplitude.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Slope	Sets the signal transition direction to detect (rising or falling).

Mode & Holdoff panel (Edge Trigger configuration menu) fields and controls

Field or control	Description
Trigger Mode	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event: **Auto** trigger mode enables the instrument to acquire and display a waveform even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.

Field or control	Description
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **User Holdoff** sets the instrument to wait for a user to trigger the another trigger event. It is available when enabled through PI command. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Act on Trigger	Tap the **Act on Trigger** button to configure the actions the instrument must take when a trigger event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT.

Pulse Width Trigger configuration menu

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

To open the Pulse Width trigger configuration menu:

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

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

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Trigger When	• **< Limit**: A pulse width is less than the specified time limit. • **> Limit**: A pulse width is greater than the specified time limit. **Inside Range**: A pulse width is in the specified time range.
Level	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Time Limit	Sets the time period condition to be met.

Field or control	Description
High Time Limit (Trigger When = Inside Range or Outside Range)	Sets the longest acceptable pulse width time period for the range condition.
Low Time Limit (Trigger When = Inside Range or Outside Range)	Sets the shortest acceptable pulse width time period for the range condition.
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Logic Qualification	Set to **On** to enable logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the Logic Qualification - Define Inputs dialog. Use this dialog to set the logic state, threshold levels, and logic operation of the input signals. Available when **Logic Qualification** is set to **On**. See Logic Qualification - Define Inputs configuration menu (on page 513).

Mode & Holdoff panel (Pulse Width Trigger configuration menu) fields and controls

Field or control	Description
Trigger Mode	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event: **Auto** trigger mode enables the instrument to acquire and display a waveform even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **User Holdoff** sets the instrument to wait for a user to trigger the another trigger event. It is available when enabled through PI command. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.

Field or control	Description
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Act on Trigger	Tap the **Act on Trigger** button to configure the actions the instrument must take when a trigger event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT.

Logic Qualification – Define Inputs configuration menu

Use this menu to set the logic state, and threshold levels to use for the trigger.

This menu is available when the trigger type is set to Pulse Width, Timeout, Runt, Window or Rise / Fall Time.

To open the Logic Qualification - Define Inputs menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Pulse Width**, **Timeout**, **Runt**, **Window** or **Rise / Fall Time**.
3. Tap Logic Qualification to **On**.
4. Tap the **Define Inputs** button.

Logic Qualification - Define Inputs (Trigger configuration menu) fields and controls

Field or control	Description
Chx (analog channels)	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. Use the Threshold field to set the signal level that must be exceeded for that signal to be true (logical 1).
Set All	Sets all signal sources to detect a logic **High**, **Low**, or **Don't Care** condition.
Define Logic	Sets the logic condition that must occur with all inputs. • **AND**: All conditions are true. • **OR**: Any condition is true. • **NAND**: One or more conditions are true. • **NOR**: No conditions are true.

Rise/Fall Time Trigger configuration menu

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

To open the Rise/Fall Time trigger configuration menu:

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

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

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Trigger When	• **< Limit**: A signal has a rise/fall time less than the specified time limit. • **> Limit**: A signal has a rise/fall time greater than the specified time limit.
Time Limit	Sets the time period condition to be met.
Slope	Sets the signal transition direction to detect (rising or falling).
Upper Threshold	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
Logic Qualification	When set to **On** enables logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the **Logic Qualification - Define Inputs** configuration menu. Available when **Logic Qualification** is **On**. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See Logic Qualification - Define Inputs configuration menu (on page 513).

Mode & Holdoff panel (Rise/Fall Time Trigger configuration menu) fields and controls

Field or control	Description
Trigger Mode	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event: **Auto** trigger mode enables the instrument to acquire and display a waveform even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **User Holdoff** sets the instrument to wait for a user to trigger the another trigger event. It is available when enabled through PI command. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Act on Trigger	Tap the **Act on Trigger** button to configure the actions the instrument must take when a trigger event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT.

Runt Trigger configuration menu

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

To open the Runt trigger configuration menu:

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

Settings panel (Runt Trigger configuration menu) fields and controls

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Trigger When	• **Occurs**: A runt signal event occurs. • **< Limit**: A runt signal event occurs that has a pulse width less than the specified time limit. • **> Limit**: A runt signal event occurs that has a pulse width greater than the specified time limit.
Polarity	Sets the polarity of the pulse to detect (positive pulse only, negative pulse only, or a positive or negative pulse).
Time Limit	Sets the time period condition to be met.
Upper Threshold	Sets the upper amplitude level through which the signal must pass to be considered a valid transition.
Lower Threshold	Sets the lower amplitude level through which the signal must pass to be considered a valid transition.
Logic Qualification	Set to On to enable logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the **Logic Qualification - Define Inputs** configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. Available when **Logic Qualification**is set to **On**. See Logic Qualification - Define Inputs configuration menu (on page 513).

Mode & Holdoff panel (Runt Trigger configuration menu) fields and controls

Field or control	Description
Trigger Mode	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event: **Auto** trigger mode enables the instrument to acquire and display a waveform even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **User Holdoff** sets the instrument to wait for a user to trigger the another trigger event. It is available when enabled through PI command. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Act on Trigger	Tap the **Act on Trigger** button to configure the actions the instrument must take when a trigger event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT.

Sequence Trigger configuration menu

Use Sequence triggers to trigger on a second (B) event after a first (A) event occurs. You can specify to trigger on the first occurrence of event B (with or without a time delay), or trigger after a specified number of B events occur.

To open the Sequence trigger configuration menu:

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

Settings panel (Sequence Trigger configuration menu) fields and controls

Field or control	Description
A Trigger Event	Tap to open the **A Trigger Event** menu to select the first (A) event trigger condition. If the A event does not occur, no trigger event is generated. The **Sequence** trigger type is not available in the A Trigger Event menu. See A Trigger Event configuration menu.
B Trigger Event	Tap to open the **B Trigger Event** menu to select the second (B) event trigger condition. If the A event occurs but the B event does not occur, no trigger event is generated. The **Sequence** trigger type is not available in the B Trigger Event menu. See B Trigger Event configuration menu (on page 520).
After the A Trigger Event is found: Trigger on the 1st B event	Sets the oscilloscope to trigger on the first occurrence of the B event trigger conditions.
After a Delay of:	Sets a time delay condition for the Trigger on 1st B event condition. The oscilloscope waits the specified time period after the A event before detecting and triggering on the B event condition. Available when **After the A Trigger Event is found** is **Trigger on the 1st B event**.
Reset Type	Resets the sequence without triggering event occurs before the B event.
After the A Trigger Event is found: Trigger on the Nth B event	Sets the B trigger event to wait for a specified number of trigger events before generating a trigger.
Where N is:	Sets the number of B trigger events that must occur before triggering the oscilloscope. Available when **After the A Trigger Event is found** is **Trigger on the Nth B Event**.
After the A Trigger Event is found: Trigger on every B Event	Sets the oscilloscope to trigger on the occurrence of the B event trigger.
After the A Trigger Event is found: Trigger on B Event Scan	Sets the oscilloscope to trigger on the occurrence of the B event scan.
B Event Scan	Tap to setup the B event scan method. Available when **After the A Trigger Event is found** is **Trigger on B Event Scan**.
B EVENT SCAN SETUP	

B Scan Method	Sets the B event scan method as **Sequential**, **Toggle**, or **Random**. ▪ **Sequential**: Progresses through each B trigger event from the start event to the end event. Each B trigger event will be triggered on N times where N is specified by the Advance After N Occurrences control. ▪ **Toggle**: Toggle between the B trigger Start event and B trigger End event. Each B trigger event will be triggered on Nth times where N specified by the Advance After N occurrences control. ▪ **Random**: Randomly picks a B trigger event in the specified range and triggers on it N times where N is specified by the Advance After N Occurrences control before proceeding to the next randomly picked B trigger event.
Start Event	Sets the start of trigger B event.
End Event	Sets the end of trigger B event.
Advance After N Occurence(s)	Sets the occurrence of trigger B event. Each B trigger event will be triggered on N times where N is specified by the advance after N occurrences control.

Mode & Holdoff panel (Sequence Trigger configuration menu) fields and controls

Field or control	Description
Trigger Mode	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event: **Auto** trigger mode enables the instrument to acquire and display a waveform even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Holdoff is not compatible with Sequence triggering.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT.

A Trigger Event configuration menu

Use this menu to set the trigger conditions for the A trigger event of a Sequence trigger.

To open the A Trigger Event menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Sequence**.
3. Tap the **A Trigger Event** button.

A Trigger Event menu (Sequence Trigger configuration menu) fields and controls

Field or control	Description
Trigger Type	Select the A event trigger type.
Other fields and controls	The fields and controls shown depend on the selected Trigger Type. Use the following links to access setting information on a specific trigger. ▪ Edge Trigger menu (on page 509) ▪ Pulse Width Trigger menu (on page 511) ▪ Rise/Fall Time Trigger menu (on page 513) ▪ Runt Trigger menu (on page 516) ▪ Sequence Trigger menu ▪ Timeout Trigger menu (on page 521) ▪ Window Trigger menu **NOTE:** The Sequence Trigger type is not available in the A Trigger Event menu.

B Trigger Event configuration menu

Use this menu to set the trigger conditions for the B trigger event of a Sequence trigger.

To open the B-Trigger Event configuration menu:

1. Double-tap the **Trigger** badge on the Settings bar.
2. Set the **Trigger Type** to **Sequence**.
3. Tap the **B Trigger Event** button.

B Trigger Event menu (Sequence Trigger configuration menu) fields and controls

Field or control	Description
Trigger Type	Select the B event trigger type.
Other fields and controls	The fields and controls shown depend on the selected Trigger Type. Use the following links to access setting information on a specific trigger. ▪ Edge Trigger menu (on page 509) ▪ Pulse Width Trigger menu (on page 511) ▪ Rise/Fall Time Trigger menu (on page 513) ▪ Runt Trigger menu (on page 516) ▪ Sequence Trigger menu ▪ Timeout Trigger menu (on page 521) ▪ Window Trigger menu **NOTE:** The Sequence Trigger type is not available in the B Trigger Event menu.

Timeout Trigger configuration menu

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

To open the Timeout trigger menu:

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

Settings panel (Timeout Trigger configuration menu) fields and controls

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.
Trigger When	• **Stays High**: The signal stays above the specified threshold level longer than the specified time. • **Stays Low**: The signal stays below the specified threshold level longer than the specified time. • **Either**: The signal stays above or below the specified threshold level longer than the specified time.
Threshold	Sets the amplitude level that the signal must pass through to be considered a valid transition.
Set to 50%	Sets the threshold at 50% of the measured signal transition range. 50% is calculated as (Top + Bottom)/2.
Time Limit	Sets the time period condition to be met.
Logic Qualification	When set to On, enables logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the **Logic Qualification - Define Inputs** configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. See Logic Qualification - Define Inputs configuration menu (on page 513). Available when Logic Qualification = On.

Mode & Holdoff panel (Timeout Trigger configuration menu) fields and controls

Field or control	Description
Trigger Mode	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event: **Auto** trigger mode enables the instrument to acquire and display a waveform even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **User Holdoff** sets the instrument to wait for a user to trigger the another trigger event. It is available when enabled through PI command. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Act on Trigger	Tap the **Act on Trigger** button to configure the actions the instrument must take when a trigger event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT.

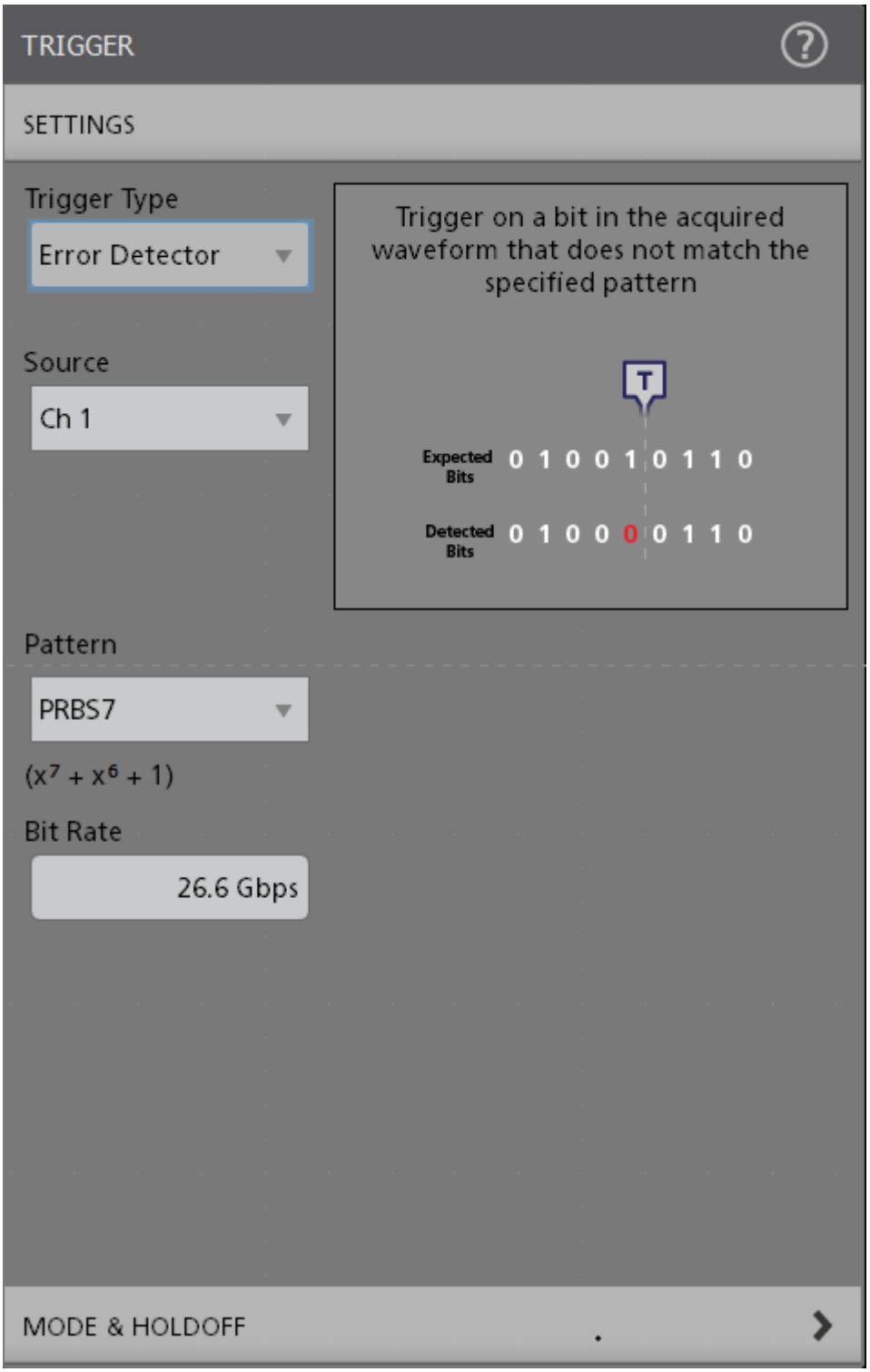
Error Detector configuration menu

Use the Error Detector trigger to trigger on bit errors detected in NRZ bit-streams with data rates between 1 Gbps and 26.6 Gbps.

To open the Error Detector trigger configuration menu:

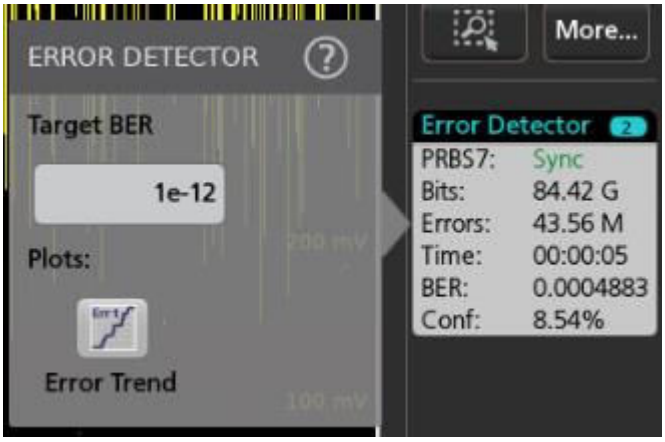
1. Double-tap the Trigger badge on the Settings bar.
2. Set the **Trigger Type** to **Error Detector.**

Settings panel (Error Detector Trigger configuration menu) fields and controls



Field or control	Description
Source	Selects the source channel for the Error Detector.
Pattern	Sets the pattern to compare incoming bits against. Select from a list of predefined PRBS Patterns (see the PRBS Patterns (on page 524) topic) or Custom (see the Custom Pattern files (on page 525) topic).
Pattern File	Available when Pattern is set to Custom. Allows you to browse for and select a custom pattern file. See the Custom Pattern files (on page 525) topic for additional information.
Bit Rate	Sets the bit rate of the incoming bit stream.

Double-tapping the Error Detector badge opens the Error Detector menu. From this menu you can add an error trend plot or specify the Target BER value. The Target BER value is used in calculating the Confidence Level of the bit error rate test.



See Using the Error Detector (on page 557) for additional information.

PRBS Patterns

A pseudorandom binary sequence (PRBS) is a deterministic stream of binary data commonly used for testing digital communication systems. Although generated algorithmically rather than randomly, a PRBS is designed to mimic the statistical properties of random data, providing a repeatable yet realistic test pattern for evaluating system performance. Each PRBS pattern is defined by a characteristic polynomial that describes the feedback taps of the linear feedback shift register (LFSR) used to generate the sequence. Commonly referenced patterns, such as PRBS7, PRBS15, PRBS23, and PRBS31, are named according to the order of their generating polynomial and produce sequences with lengths of 2n-1 bits before repeating. The following table lists the standard PRBS patterns available in the Error Detector and the corresponding polynomials used to generate them.

Pattern	Polynomial
PRBS7	$(x^7 + x^6 + 1)$
PRBS9	$(x^9 + x^5 + 1)$
PRBS11	$(x^{11} + x^9 + 1)$
PRBS13	$(x^{13} + x^{12} + x^2 + 1)$
PRBS15	$(x^{15} + x^{14} + 1)$

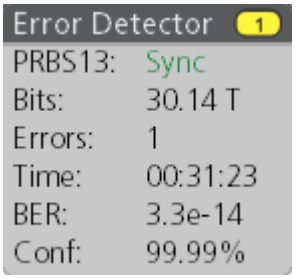
PRBS16	$(x^{16} + x^{15} + x^{13} + x^4 + 1)$
PRBS20	$(x^{20} + x^{17} + 1)$
PRBS21	$(x^{21} + x^{19} + 1)$
PRBS23	$(x^{23} + x^{18} + 1)$
PRBS31	$(x^{31} + x^{28} + 1)$

Custom Pattern files

Custom user-defined pattern files can be used to set the pattern the Error Detector will compare the incoming bit-stream. These pattern files are defined using a simple text file containing a pattern of 1's and 0's. Valid pattern files must meet the following requirements.

- ASCII text file (.txt) containing only '1' and '0' characters and white space characters.
- Maximum pattern length of 216 x 80 bits (5.2 Mbit).
- The pattern length must be a multiple of 80 bits.
  - If the length of the pattern is not a multiple of 80 bits, the pattern will automatically be duplicated and concatenated enough times to produce a total pattern length that is a multiple of 80 bits.
  - If duplicating the pattern causes the pattern length to exceed the maximum pattern length limit, an out-of-memory error will be reported.
- The first 80-bits of the pattern must be unique. This ensures reliable Error Detector synchronization.

Error Detector results



When the Error Detector is running, Error Detector status and results will be displayed in the Error Detector badge.

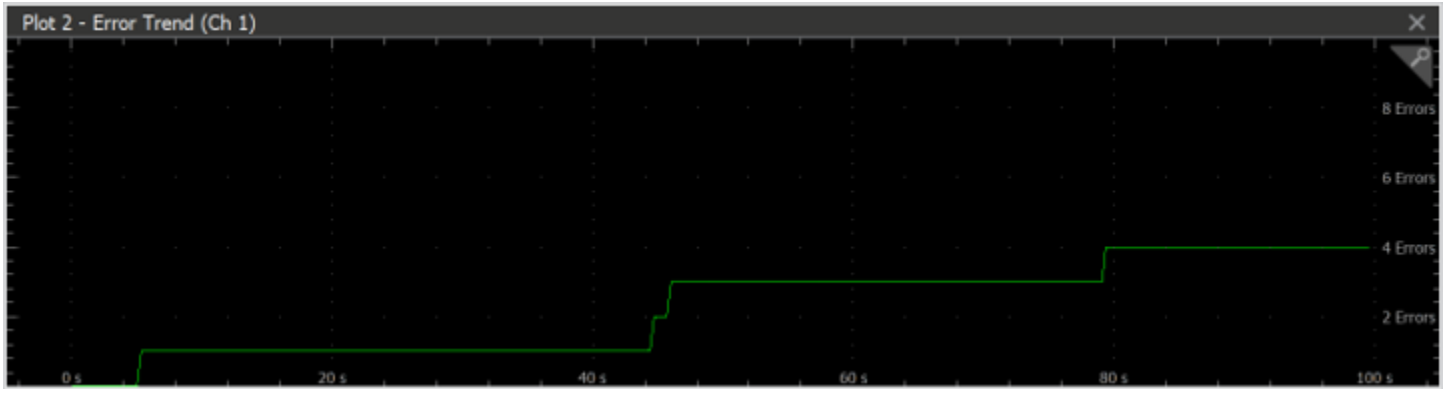
Results field	Description
Pattern Name	Displays the selected pattern which the incoming bit stream is being compared against by the Error Detector.
Error Detector Status	Displays the status of the Error Detector. Status can have the following values. Sync: Clock data recovery is locked to the incoming signal, and the Error Detector is aligned with the incoming data pattern. **No Sync**: Clock data recovery is locked to the incoming signal. However, the Error Detector could not align to the incoming data pattern. **No Lock**: Clock data recovery is not locked to the incoming signal.
Bits	The total number of bits tested.

Errors	The total number of bit errors found.
Time	The total time the Error Detector has been running.
BER	The calculated Bit Error Rate. BER = Errors / Bits.
Conf	Calculated confidence level based on the number of bits acquired, the number of errors, and the user-specified Target BER. The reported range is 0% to 100%. Confidence Level is a statistical measure that tells you the probability that a system's true error rate is below a specified target value. Because error testing takes finite time, finding zero errors in a test only guarantees performance to a certain statistical certainty. Increasing the amount of time the Bit Error Rate Test runs error free increases the confidence level.

Error Trend Plot

To add an error trend plot:

1. Double-tap the Error Detector badge in the Results Bar.
2. Tap the **Error Trend** button to add the Error Trend plot.



The Error Trend Plot displays the number of errors that have occurred during the Bit Error Rate test. The X-axis displays the elapsed time in seconds since the Error Detector started. The Y-axis displays the total Error Count.

Window Trigger configuration menu

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

To open the Window trigger menu:

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

Settings panel (Window Trigger configuration menu) fields and controls

Field or control	Description
Source	Lists the source channel or waveform to use to trigger or search. Types that require multiple inputs will replace this control with a different source definition control.

Trigger When	• **Enters Window**: The signal outside a window enters the window defined by the upper and lower threshold settings. • **Exits Window**: The signal exits the window defined by the upper and lower threshold settings. • **Inside > Limit**: The signal remains inside the window longer then the specified time limit. • **Outside > Limit**: The signal remains outside the window longer then the specified time limit.
Upper Threshold	Sets the amplitude threshold value for the upper edge of the window.
Lower Threshold	Sets the amplitude threshold value for the lower edge of the window.
Time Limit	Sets the time period condition to be met. Available when Trigger When = Inside > Limit or Outside > Limit.
Threshold Crossing (Trigger When = Outside > Limit)	• **Upper**: A signal remains above the upper threshold level for longer than the specified time limit before crossing the upper threshold level to a lower level. • **Lower**: A signal remains below the lower threshold level for longer than the specified time limit before crossing the lower threshold level to a higher level. • **Either**: A signal remains outside (above or below) the two threshold levels for longer than the specified time limit before crossing either threshold level. • **None**: A signal remains outside the two specified threshold levels for longer than a specified time limit.
Threshold Crossing (Trigger When = Inside > Limit)	• **Upper**: A signal remains between two thresholds for longer than the specified time limit before crossing through the upper threshold. • **Lower**: A signal remains between two thresholds for longer than the specified time limit before crossing through the lower threshold. • **Either**: A signal remains between two thresholds for longer than the specified time limit before crossing through either the upper or lower threshold. • **None**: A signal remains between two threshold levels for longer than a specified time limit.
Logic Qualification	When set to On, enables logic qualification to further refine the trigger condition by setting the required logic conditions on the source signals to generate a trigger event.
Define Inputs	Opens the **Logic Qualification - Define Inputs** configuration menu. Use this menu to set the logic state, threshold levels, and logic operation of the input signals. Available when Logic Qualification is set to **On**. See Logic Qualification - Define Inputs configuration menu (on page 513).

Mode & Holdoff panel (Window Trigger configuration menu) fields and controls

Field or control	Description
Trigger Mode	The trigger mode determines how the instrument behaves in the absence or presence of a trigger event: **Auto** trigger mode enables the instrument to acquire and display a waveform even if a trigger does not occur. Auto mode uses a timer that starts when the acquisition is started, and the pretrigger information is obtained. If a trigger event is not detected before the timer times out, the instrument forces a trigger. The length of time it waits for a trigger event depends on the time base setting. When forcing triggers in the absence of valid triggering events, Auto mode does not synchronize the waveform on the display. The waveform will appear to jump across the screen. If valid triggers occur, the display will become stable. **Normal** trigger mode enables the instrument to acquire a waveform only when it is triggered. If no trigger occurs, the last waveform record acquired remains on the display. If no last waveform exists, no waveform is displayed.
Force Trigger	Forces a trigger event regardless of whether the waveform meets any trigger conditions.
Holdoff	Trigger holdoff sets the amount of time the oscilloscope waits after a trigger event before detecting and triggering on the next trigger event. **User Holdoff** sets the instrument to wait for a user to trigger the another trigger event. It is available when enabled through PI command. **Random** sets the instrument to wait for a random amount of time before recognizing another trigger event. This means that successive acquisitions are unrelated to the previous trigger signal. **Time** sets the instrument to wait the specified time before recognizing another trigger event. Use this option when the signal that you want to trigger on has several possible trigger points or is a burst signal.
Holdoff Time	Tap the **Holdoff Time** field and use the multipurpose knob to adjust the holdoff time value. Or double-tap the field and use the virtual keypad to enter a time holdoff period.
Act on Trigger	Tap the **Act on Trigger** button to configure the actions the instrument must take when a trigger event occurs. See Act On Event configuration menu (on page 163) for the available fields and controls.
Trigger Frequency Counter	Turn **On** to display the trigger event frequency in the **Trigger** badge. The trigger frequency can help you troubleshoot signal problems where the frequency of the trigger event may be related to a clock, switching power supply, or other recurrent frequency that occurs on your DUT.

Text Settings configuration menu (Callout and Waveform labels text)

Use this menu to change and format existing Callout or Waveform labels (font type and size, color, bold, italic, and underline).

Prerequisite: There is callout or waveform label on the screen.

To open the **Text Settings** configuration menu, double-tap on the Callout or Waveform label. You can also touch and hold on the Callout/label text and select **Format Text** from the right-click menu.

To delete a Callout or label, touch and hold on the Callout/label and select **Delete** from the right-click menu.

Text Settings configuration menu fields and controls

Field or control	Description
Text	Double-tap and enter the desired text using the virtual keyboard.
Font	Tap and select the desired font from the drop-down list.
Size	Tap and select the desired font size from the drop-down list.
Color	Tap and select the desired font color from the color palette.
B	Tap to toggle text bolding On or Off.
I	Tap to toggle text italics On or Off.
U	Tap to toggle text underlining On or Off.

Virtual Keyboard

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

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

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

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

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

Virtual Keypad

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

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

Virtual Keypad fields and controls

Field or control	Description
Clear	Clears all values from the input entry field.
Exp	Lets you enter exponential notation entries.
Max	Enters the maximum value allowed for this setting.
Min	Enters the minimum value allowed for this setting.
Bksp ←	Deletes characters to the left of the insert text marker position.

Field or control	Description
Enter	Closes the number pad and assigns the entered value to the field.
±	Tap the button to set a numeric value to a positive (default) or negative value.
Unit buttons	Use to set the units of the entered value.

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
Visual Trigger	Enables or disables Visual Trigger mode.
Display	**Areas**: When this is checked, draws the visual trigger areas in the Waveform View. **Criteria**: 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)). **Logic Equation**: 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. When zoom is on, the equation is in the upper left of the Zoom Overview area. To create or edit the area combinatorial logic equation, double tap on the equation to open the **Area Combinatorial Logic** keypad. Note that you can only open the keypad if more than one visual trigger area is defined. See Visual Trigger Area Combinatorial Logic menu (on page 532)

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 CreateVisualTriggerAreas-6.xml (on page 544)

Area Settings panel fields and controls

Field or control	Description
Source	Lists the available source signals for which to create an area. The default value is the signal source of the selected area.

Field or control	Description
Shape	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.
Waveform Must Be	Sets the logic condition for how the area logic equation interprets when a waveform intersects an area. **In (✓)**: A waveform must intersect this area for the area logic to be true. **Out (X)**: A waveform must not intersect this area for the area logic to be true. **Don't Care (?)**: 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.
Height	Sets the area height, in amplitude units, between the topmost vertex and the bottommost vertex.
Vertical Center	Sets the area vertical center, in amplitude units, as the point halfway between the topmost vertex and bottommost vertex.
Flip Vertical	Flips the area vertically around its Vertical Center value.
Width	Sets the area width, in time units, between the leftmost vertex and the rightmost vertex.
Horizontal Center	Sets the area horizontal center, as time units, as the point halfway between the leftmost vertex and rightmost vertex.
Flip Horizontal	Flips the area horizontally around its Horizontal Center value.
Rotate	Rotates the area in units of degrees, from 0° to 360°. 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°. The area height and width are relative to the current area orientation, and automatically change places as needed when the area is rotated.

Edit Vertices panel fields and controls

Field or control	Description
Define Area	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. Selecting a row in the table highlights the associated vertex on the area. Use the Multipurpose knobs to change the value of the X or Y settings, or double-tap a setting and enter the value directly.

Field or control	Description
Insert Point	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.
Delete Point	Deletes the currently selected point, keeps the row selected, and moves all rows below it up one row. The **Delete Point** button is not available when a triangular area is selected.
Reset Points	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.

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 532).

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 **In**.
Out	Sets the area's Waveform Must Be control to **Out**.
Don't Care	Sets the area's Waveform Must Be control to **Don't Care**.
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.

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.

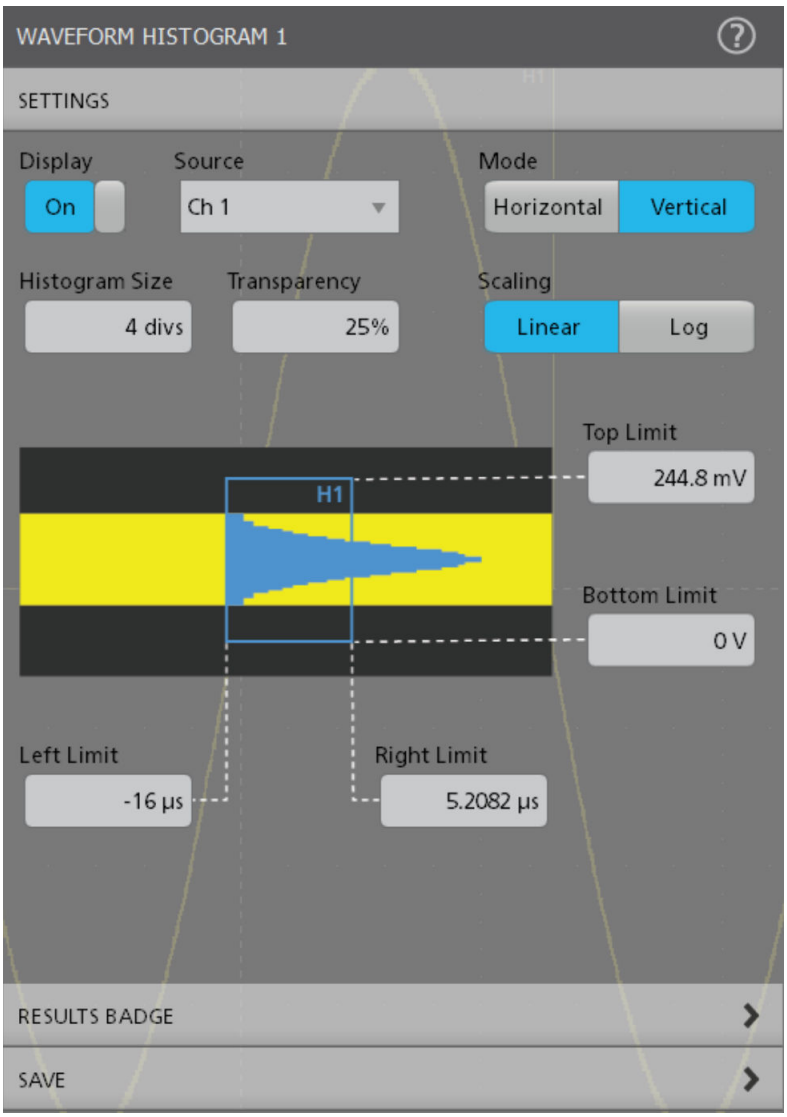
Area Combinatorial Logic menu fields and controls

Field or control	Description
Logic expression field	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. 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. You must tap the **Enter** button to check and save the logic expression to make it valid.
Clear	Deletes all text in the logic expression field.
Move insertion point	Moves the insertion point left or right in the logic expression field.
Logic action buttons	Inserts the selected logic function or grouping parenthesis at the insertion point in the logic expression field.
Bksp	Deletes the character to the left of the insertion point.
Insert Area buttons	Inserts the selected area text into the logic expression. Only available areas are listed. If more than nine areas are defined, use the scroll bar to view area buttons greater than A9.
Cancel	Closes the menu without saving any changes.
Enter	Checks the syntax of the logic expression. If it is valid, the changes are saved and the menu is closed. 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.

Waveform Histogram configuration menu

Use the Waveform Histogram menu to edit waveform histogram parameters. Double tapping a histogram opens the Waveform Histogram configuration menu.

To open the Waveform Histogram configuration menu, double-tap on a histogram.



To create a histogram, see Create a waveform histogram (on page 149).

Field or control	Description
Display	Enables or disables displaying the waveform histogram bins.
Source	Lists the source channel or waveform for the waveform histogram.
Mode	Set the mode of the histogram as **Vertical** or **Horizontal**.
Histogram Size	Set the size of the histogram in tenths of divisions.
Transparency	Set the transparency of the histogram.
Scaling	Set the scaling of histogram as **Linear** or **Log**. Default is linear.
Histogram box limits **Top Limit**, **Bottom Limit**, **Left Limit**, **Right Limit**	Tap the field and use Multipurpose knob A and B to set a histogram box limits. To enter a specific value, double-tap on the field and use the virtual keypad to enter a limit.

Waveform Histogram Result Badge menu

The following are the histogram statistics you can add to the waveform histogram badge.

Field or control	Description
Display	Toggles the display to **On** or **Off**. Default is on. When it is on, the histogram badge displays in the Results bar.
Waveform Count	Waveform Count is the number of waveforms that contributed to the histogram.
Hits in Box	Hits in box is the number of samples within the histogram box or on its boundaries.
Peak Hits	Peak Hits is the number of samples in the bin that contains the most hits.
Maximum	Maximum is the amplitude of the highest non-zero bin in vertical histograms or the time of the rightmost non-zero bin in horizontal histograms.
Minimum	Minimum is the amplitude of the lowest non-zero bin in vertical histograms or the time of the leftmost non-zero bin in horizontal histograms.
Mean	Mean is the average of all histogram data points within or on the histogram box.
Peak-to-Peak	For vertical histograms, Peak-to-Peak is the amplitude of the highest non-zero bin minus the amplitude of the lowest non-zero bin. For horizontal histograms, it is the time of the rightmost non-zero bin minus the time of the leftmost non-zero bin.
Median	Median is the middle histogram data value, where half of all histogram data points are less than this value and half are greater than this value.
Mode	Mode is the histogram data value that occurs most frequently.
Standard Deviation	Standard Deviation is the standard deviation (Root Mean Square deviation) of all histogram data points within or on the histogram box.
μ±1σ	μ ±1σ is the percentage of hits in the histogram that are within one standard deviation of the histogram mean.
μ±2σ	μ ±2σ is the percentage of hits in the histogram that are within two standard deviations of the histogram mean.
μ±3σ	μ ±3σ is the percentage of hits in the histogram that are within three standard deviations of the histogram mean.

Save As configuration menu (Save Waveform Histogram)

Use the Save As configuration menu to set the location and file name to which to save the contents of a histogram, as a .csv format file.

To open the Save As configuration menu to save a histogram

1. Double-tap anywhere on the histogram to open the Waveform Histogram configuration menu.
2. Tap **Save > Save Histogram Data**.

Save As menu, Histogram Save fields and controls

Field or control	Description
Save Location	Sets the location of the save file. The default value is the last location to which a file was saved. Tap on the file path and use a keyboard to enter a new save location Or double-tap on the file name to open the 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.
Browse	Tap to open the Browse Save As Location dialog, to navigate to and select the location to save the file to. See Browse Save As Location configuration menu (on page 318).
File Name	Sets the file name of the file. The default file name is Tek000. The file name increments on the next save (Tek001, Tek002, and so on). Tap on the file name and use a keyboard to enter a new file name. Or double-tap on the filename to open the virtual keyboard and enter a file name.
Save as Type	Save the histogram as a .csv file.

Right click menu functions associated with the Waveform Histogram

The following functions are available when you right click on a waveform histogram.

Field or control	Description
Configure Histogram	Opens the histogram configuration menu for the selected histogram.
Delete Histogram	Deletes the selected histogram.
Delete All Histograms	Deletes all the histograms.

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

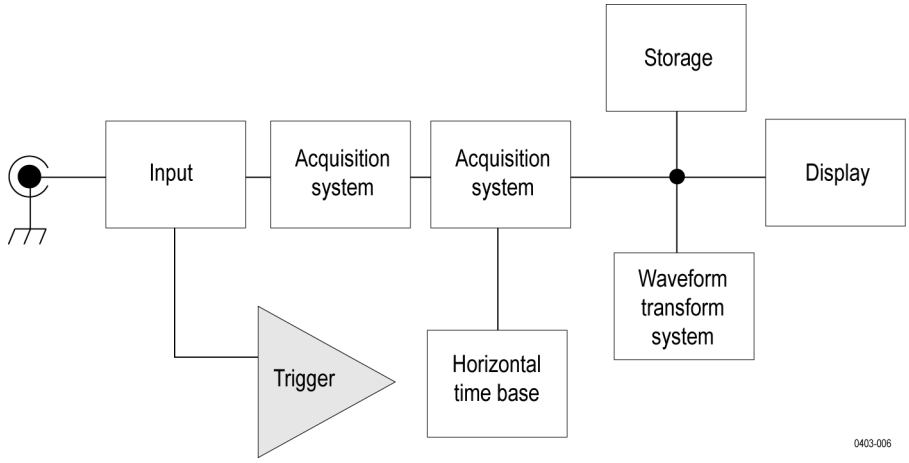
Fields or controls	Description
Display Mode	Sets how waveforms are shown on the screen. **Overlay** mode displays all waveforms on the screen, overlaid on each other. This is the traditional way that oscilloscopes displayed waveforms, and lets you overlay waveforms to do direct comparisons of waveform shape. The grid vertical position and scale values change for each selected waveform. **Stacked** mode draws each waveform in an individual section, or slice, of the screen, stacked one on top of the other. This lets you view each waveform uncluttered by overlaying waveforms. Each waveform slice displays its own vertical scale units. A trigger level indicator in a slice indicates that that waveform is the trigger source. **NOTE:** Grouping badges requires the display mode be set to **Stacked** 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. See Group signal badges in the Settings bar (on page 56).
Interpolation	Selects the method used to display record points between sampled points. **Sin(x)/x** computes record points along a curve between the actual acquired samples. This form of interpolation is useful when acquiring rounded waveforms such as sine waves. It is good for general-purpose uses but may introduce overshoot or undershoot in signals with fast rise times. This interpolation is also useful for looking at high-frequency signals, especially where the frequency components are just below the Nyquist frequency. **Linear** computes record points between actual acquired samples using a straight-line fit. This interpolation is useful for measuring waveforms with fast rise times, such as pulse trains.
Persistence	Sets the length of time data points are displayed on screen before being erased. **Off** sets the record points to appear for the current acquisition only. **Infinite** continuously accumulates record points on the waveform until you change one of the acquisition display settings or clear the acquisition memory. Use infinite persistence for displaying record points that may occur outside the normal acquisition envelope. **Variable** lets you specify a time length to retain data points on screen. Each record point decays independently according to the time interval. **Auto** sets the **Waveform Intensity** field to control the persistence time.
Variable Persistence Time (Persistence = Variable)	Sets the length of time data points remain displayed. Tap the field and use the A knob to adjust, or double-tap and use the keypad to set a time.
Waveform Style	Sets how waveforms are drawn on the screen. **Vectors** draws waveforms with lines between record points. **Dots** draws waveform record points as dots on the screen, and adds crosshair markers to real sampled points.
Waveform Intensity	Sets the brightness of the waveform. Tap the field and use the A knob to set the waveform intensity.

Fields or controls	Description
Graticule Style	Sets the type of graticule to display. **Grid** shows a traditional grid on the instrument display. This is the default grid mode. **Time** displays vertical graticules for just the horizontal (time) scale units. **Full** displays the same style graticule lines for both horizontal and vertical scale units. **None** turns off the graticule, including the vertical scale readouts.
Graticule Intensity	Sets the brightness of the graticule. Tap the field and use the A knob to set the graticule intensity.
Plots	Displays the available plots for the selected analysis. Click **Plots** to configure plot setting and view plot results. For more information, see Plot configuration menus.

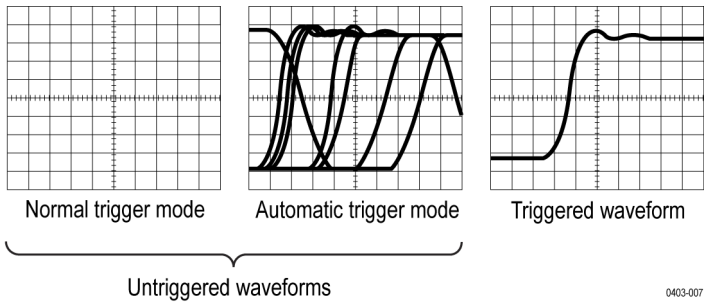
Triggering concepts

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

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



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



The trigger event

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

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

Trigger sources

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

- **Input channels**: Analog input channels are the most commonly used trigger sources. You can select any of the input channels. The channel that you select as a trigger source will function whether it is displayed or not.

Trigger types

The available trigger types include:

Cycle: Use cycle trigger to trigger on a specific cycle. It can detect any cycle within limits or those that are inside or outside a specified time range.

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.

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

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

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

Sequence: Use the A Trigger Event with the B Trigger Event to capture complex data.

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

Error Detector: Trigger on bit errors detected for NRZ on data rates between 1 Gbps and 26.6 Gbps. See the Error Detector configuration menu (on page 522) topic for additional information.

Window: Use the Window trigger to trigger when the input signal is 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.

Trigger modes

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

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

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

Trigger 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. 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.

Learn about Trigger coupling (on page)

Learn about Trigger modes (on page 540)

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.

Learn about trigger position (on page 540)

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.

Cycle trigger concepts

A cycle trigger occurs when the instrument detects a cycle that is within limits or inside or outside specified time range.

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.

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.

Runt trigger concepts

A runt trigger occurs when the instrument detects a short pulse that crosses one threshold but fails to cross a second threshold before recrossing the first.

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.

Timeout trigger concepts

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

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.

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 areas is associated with a specific analog input channel, and you can define areas for multiple channels. See Create Visual Trigger areas (on page 544). 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 546).

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 546).

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

- 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.

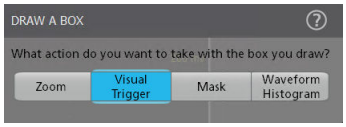
Advanced triggering

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.

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 **More** button and select the **Visual trigger** button on the **DRAW A BOX** menu window. This sets the Draw-a-Box function to Visual trigger mode.



3. 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.
4. Tap the Draw-A-Box **Zoom icon** at the up right of the **Results Bar** to 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).
5. 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.
6. When you are done drawing visual trigger areas, single-tap anywhere to end the area draw function.
7. To draw areas again, tap the Draw-A-Box **zoom** icon again. Single tap allow you to draw the box once following which the Draw-A-Box is reset automatically. Whereas double tap allow you to draw boxes multiple times continuously.

NOTE: The draw box button autoselects if you choose an option in the More context panel.

8. To change the shape of an area, see Edit visual trigger areas on the screen (on page 544) and Edit visual trigger areas using the Area menu (on page 546).

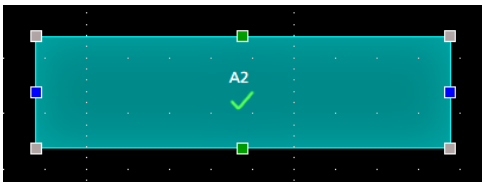
Edit visual trigger areas on the screen

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.

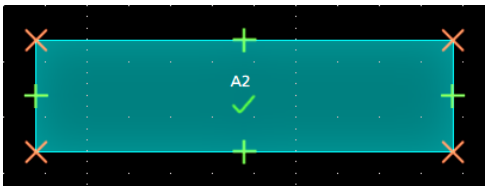
2. To delete a visual trigger area:
   a. Touch and Hold or right-click on the area to delete.
   b. Select **Delete Area** from the menu.
3. To resize a visual trigger area:
   a. Single-tap an area to enable resize area mode. Resize mode draws resize points on all four sides and the corners of a boundary 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 add vertices to or delete vertices from a visual trigger area:
   a. Tap an area twice 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.
5. To move vertices in a visual trigger area:
   a. Tap the area three times to enable move vertex mode. The move vertex mode draws crosshairs on all vertices of the area.

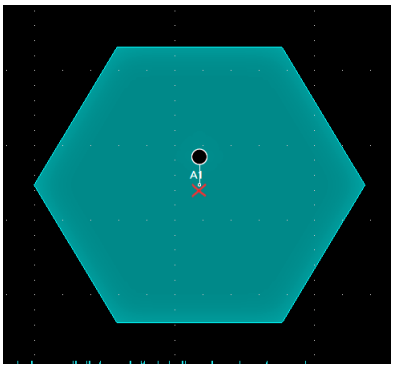


b. Touch and drag a vertex to move it to a new position.

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

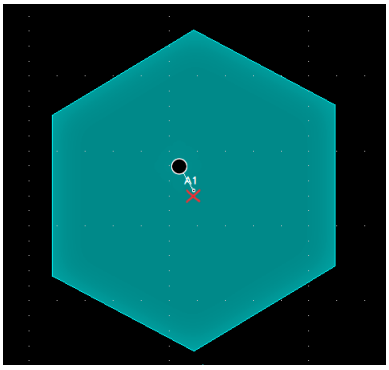
6. To rotate a visual trigger area:

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 530) for more information on the Area menu.

See Edit visual trigger areas on the screen (on page 544) 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:

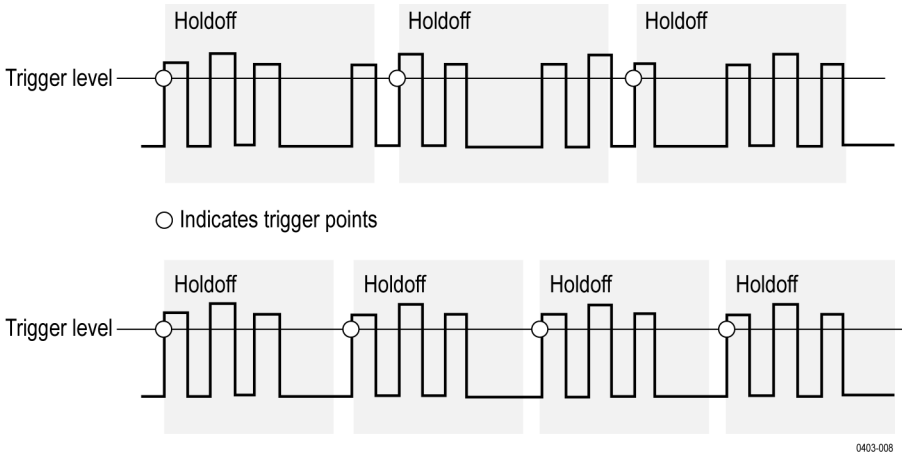
a. Double-tap on the area to edit.
b. Tap the **Flip Vertical** button. The area immediately flips vertically around its center position.
c. Tap outside the menu to exit.

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 deleted 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.

Set a Trigger Holdoff

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

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



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

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

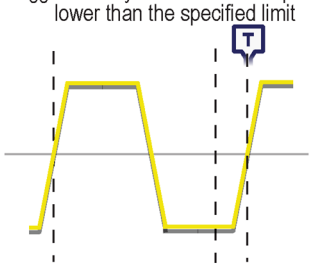
Trigger on a cycle event

Cycle triggering triggers the oscilloscope when a signal cycle is less than, greater than, equal to, or not equal to a specified limits.

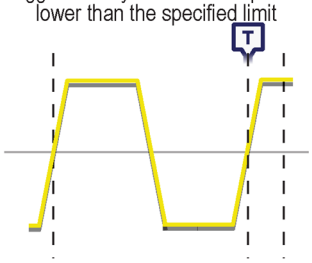
To trigger on a cycle event:

1. Double-tap the **Trigger** badge to open the Trigger configuration menu.
2. Tap **Trigger Type** and select **Cycle**. See Cycle Trigger configuration menu (on page 507).
3. Tap **Source** and select the trigger source.
4. Tap **Trigger When** and select the cycle on which to trigger (> Limit, < Limit, = Limit, ≠ Limit, Outside Range, and Inside Range).

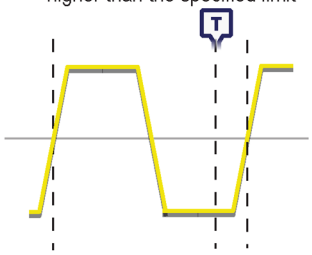
Trigger on a cycle that has a frequency lower than the specified limit



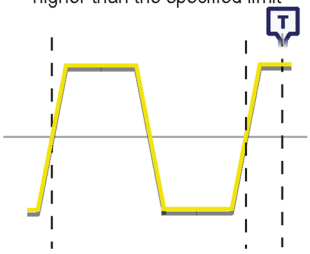
Trigger on a cycle that has a period lower than the specified limit



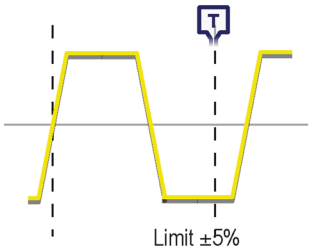
Trigger on a cycle that has a frequency higher than the specified limit



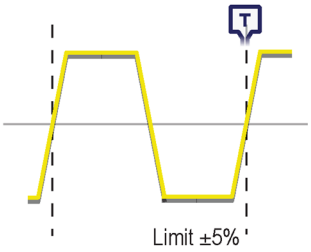
Trigger on a cycle that has a period higher than the specified limit



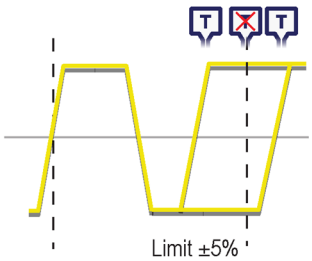
Trigger on a cycle that has a frequency equal to the specified limit



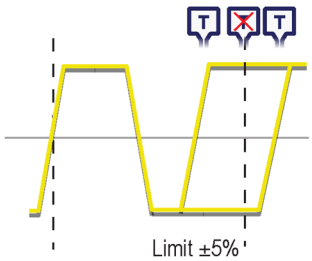
Trigger on a cycle that has a period equal to the specified limit



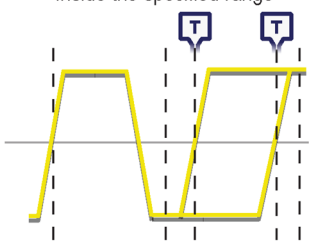
Trigger on a cycle that has a frequency not equal to the specified limit



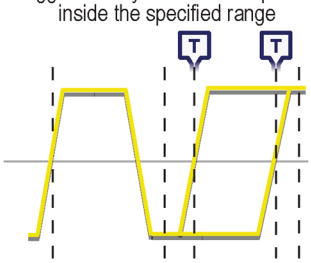
Trigger on a cycle that has a period not equal to the specified limit



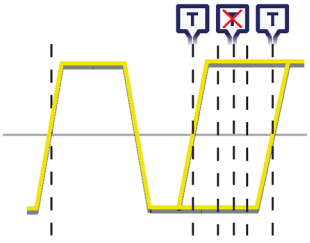
Trigger on a cycle that has a frequency inside the specified range



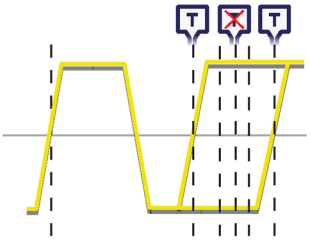
Trigger on a cycle that has a period inside the specified range



Trigger on a cycle that has a frequency outside the specified range



Trigger on a cycle that has a period outside the specified range



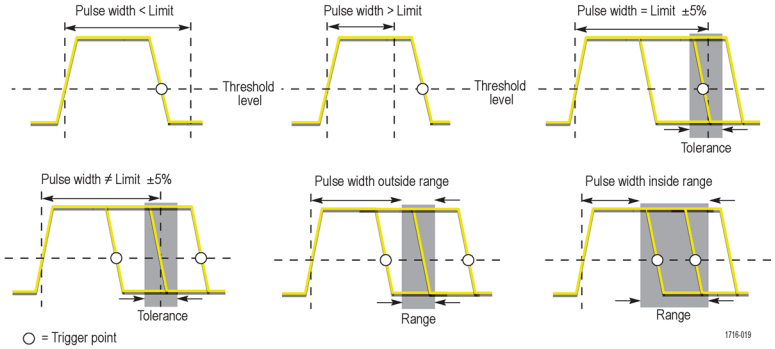
5. Tap **Level** and set the threshold value at which you want to measure cycle period or frequency.
6. Select the pulse polarity on which to trigger.
7. To further refine the trigger conditions, set **Logic Qualification** to **On** and tap the **Define inputs** button to define the logic condition 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 513).

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 511).
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, and Inside Range).



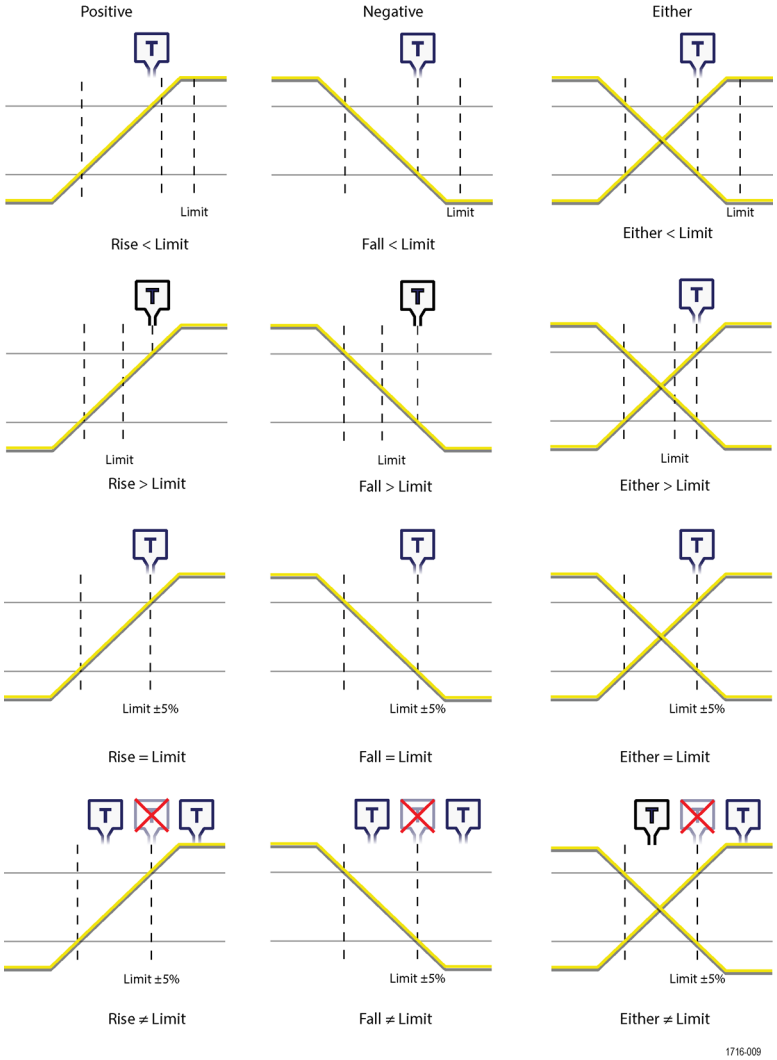
5. Set the pulse width time constraints:
   a. For all trigger when conditions except Outside Range or Inside Range, tap the **Time Limit** field and use the assigned multipurpose knob to set the pulse width time condition to meet.
   b. For Outside Range or Inside Range conditions, tap the **High Time 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 513).

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 513).
3. Tap **Source** and select the trigger signal source.
4. Tap **Trigger When** and select the condition on which to trigger.



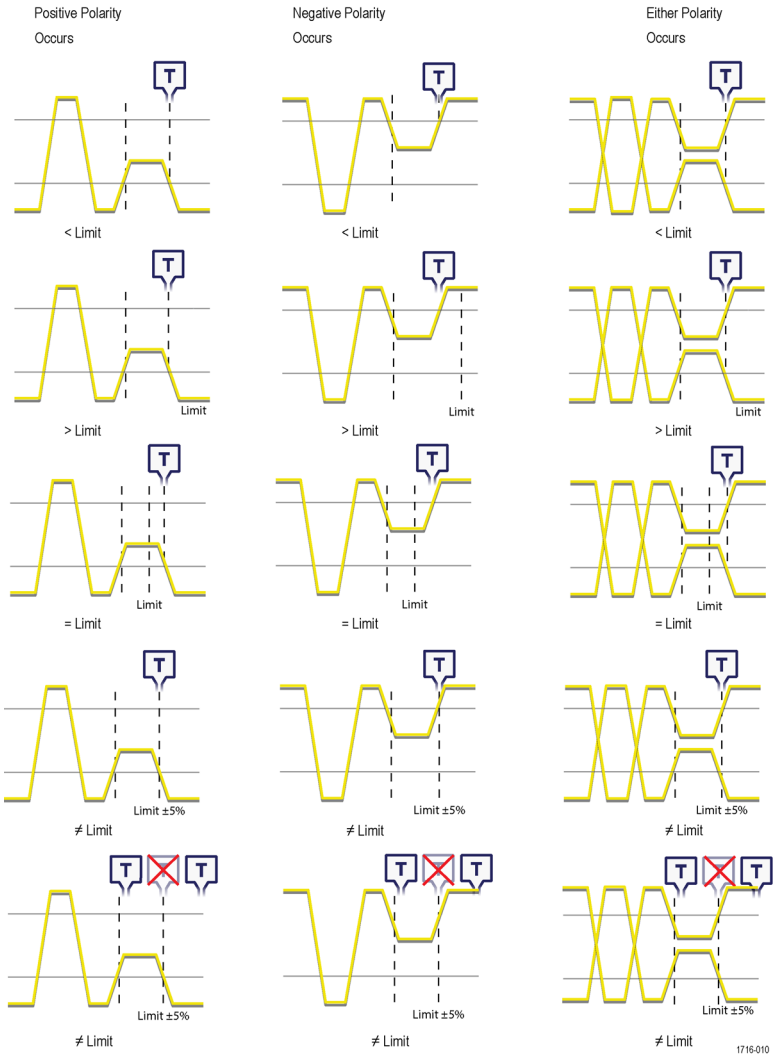
5. Tap the **Time Limit** field to set the time value for the **Trigger When** condition.
6. Tap the **Upper Threshold** field and use the assigned multipurpose knob (or double-tap the field and use the keypad to enter a value) to set the length of setup time that the data signal must be stable before 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 513).

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 516).
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 513).

Trigger on a 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
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** and select **Trigger on the Nth Trigger Event** 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** and select **Trigger on the 1st B Event** in the main Trigger menu.
   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.
7. To trigger on every B event:
   a. Tap **After the A Trigger Event is found** and select **Trigger on every B Event** in the main Trigger menu.
8. To trigger on B event scan:
   a. Tap **After the A Trigger Event is found** and select **Trigger on B Event Scan** in the main Trigger menu.
   b. Tap **B Event Scan**.
   c. Select the **B Scan Method** as **Sequential**, **Toggle**, or **Random** from the B Scan Method Setup menu.
   d. Enter the **Start Event**, **End Event**, **Advance After N occurrences(s)**. Use the multipurpose knob to set the desired values. You can also double-tap the field and use the virtual keypad to enter the values.

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:

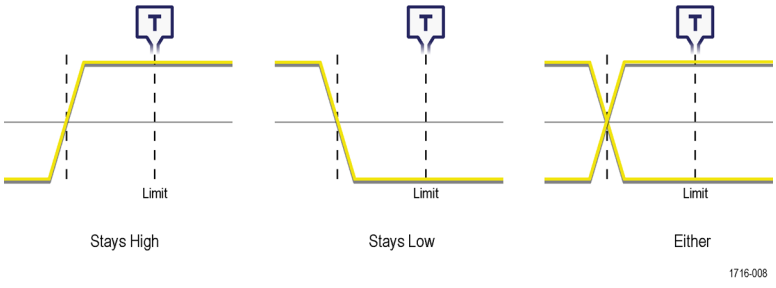
1. Double-tap the **Trigger** badge to open the **Trigger** configuration menu.
2. Tap **Trigger Type** and select **Window**. See Window Trigger configuration menu.
3. Tap **Source** and select the signal source.
4. Tap **Trigger When** and select the condition on which to trigger.
5. Tap the **Time 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 513).

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 521).
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 513).

Using the Error Detector

The Error Detector feature allows for high speed serial bit error detection and requires option 7-ERRDET and 7-ST1. The Error Detector allows you to trigger on NRZ bit streams with data rates between 1 Gbps and 26.6 Gbps for NRZ only. This simple detector uses the high-speed serial trigger system hardware to detect bit errors on a repeating pattern being sent by a serial transmitter. A bit pattern file is used to define the expected incoming pattern. Many common PRBS patterns are predefined and available from the Pattern Selection drop-down, but you can also create your own unique bit pattern files.

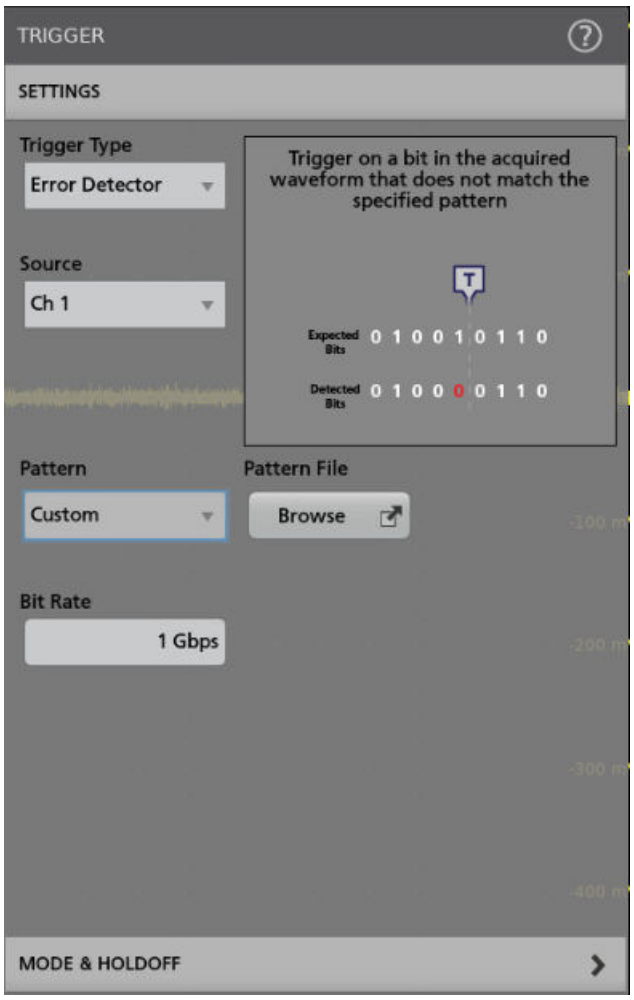
When a bit error is detected, the oscilloscope triggers a waveform acquisition, resulting in a capture of the waveform containing the bit error. The bit error that triggered the acquisitions will be aligned with the trigger position indicator within the waveform graticule. If the oscilloscope also has an optional serial decoder for the serial data stream being tested (for example, 8b10b), the acquired waveform can include decoded data, making analysis and debug of bit errors easier. You can view statistics on bits counted and number of errors detected, as well as see waveform data upon detection of a bit error.

You can view statistics on bits counted and number of errors detected, as well as see waveform data upon detection of a bit error.

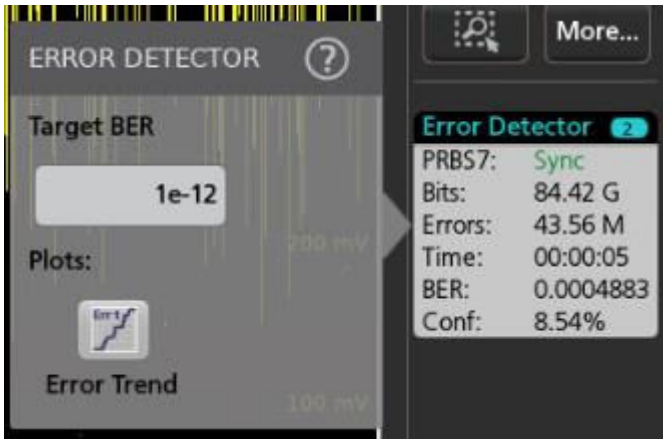
NOTE: This error detector is not protocol aware and does not detect frame/symbol/character errors.

To use the bit error detector

1. Double-tap the Trigger badge on the Settings bar to open the Trigger menu.
2. In the **Settings** group, set the **Trigger Type** to **Error Detector.**
3. Set the **Source** to the channel the incoming bit-stream is connected.
4. Set the **Pattern** to one of the predefined patterns or select **Custom.**
5. If you selected **Custom** from the Pattern drop down list, then the **Pattern File** field will appear. Browse to and select your desired pattern file.
6. Set the **Bit Rate** to match the bit rate of the incoming bit stream.



7. Tap the **Mode & Holdoff** group and change the **Trigger Mode** to Normal to disable Auto trigger. This ensures that an acquisition is triggered only when a bit error occurs.
8. Double-tap the **Error Detector** badge to open the Error Detector menu.



9. Specify the **Target BER** value. This will be used in determining the Confidence Level in the Error Detector Results.
10. Optionally, click the **Error Trend** button to add an error trend plot.
11. Press the **Run/Stop** button to start/stop the Error Detector.

Additional Information on using the Error Detector

- The acquisition Run/Stop controls are used to start/stop the Error Detector Bit Error Rate test.
- When running, the Error Detector will count the bits in the bit stream and the number of detected errors and display them in the Error Detector results.
- When a running Bit Error Rate Test is stopped, the Error Detector will stop counting bits and errors, but it will remain synchronized with the bit stream. Pressing Run will resume counting bits and errors. Bit errors that occur while the Error Detector is stopped will be ignored and will not be added to the counts when the Error Detector is started again.
- Pressing the Single Button will start the Bit Error Rate test running and stop after the configured number of Single Sequence acquisitions has been reached.
- Before the Error Detector can start counting bits, it must first be initialized. During initialization the Error Detector clock data recovery will lock onto the signal and then the Error Detector will synchronize with the incoming bit pattern. If initialization is completed successfully, the Error Detector results will display Sync. If either of these operations are not successful either a No Lock or No Sync message will be displayed.
- The following actions will cause the Error Detector to reinitialize:
  - Adding/Removing Channels
  - Changing Acquisition Settings
  - Changing Horizontal Settings
  - Changing Trigger Settings
  - Changing Vertical Settings
  - Running Autoset
  - Pressing the Single Button
  - Pressing the Clear button
- If the signal is removed from the Error Detector input, the Error Detector will lose Sync and Lock with the bit stream. If the signal is reapplied, the Error Detector may regain Lock but it will not re-sync with the pattern. Re-initialize the Error Detector using one of the actions listed above to force a relock and resync.

See the Error Detector configuration menu (on page 522) topic for additional information.

Trigger on an external signal using the AUX input

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

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

   NOTE: The **Aux** setting is available only 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.

Measurement configuration

After adding a measurement, you can customize the measurement for more precise results.

To customize measurements, double-tap a Measurement badge in the Results bar to open the measurement configuration menu (on page 300).

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 300).

Prerequisite: To set measurement reference levels you must be taking a measurement. See Add a measurement (on page 80).

1. Double-tap a **Measurement** badge in the Results bar to open the Measurement configuration menu (on page 300).
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 300).

Prerequisite: To set measurement reference levels you must be taking a measurement. See Add a measurement (on page 80).

Procedure

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 300).

To set measurement gates you must be taking a measurement. See Add a measurement (on page 80).

Procedure

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.
   - **Search** takes measurements between the start and stop times defined by a specified search.
   - **Time** takes measurements between the times specified in the **Start Gate Time** and **End Gate Time** fields, relative to the trigger point.
5. For the Logic and Search gates, tap the **Source** field and select the source from the list.
6. For the Logic gate, tap the **Threshold** field and use the assigned multipurpose knob to set the desired threshold dividing a logic Low or High state.
7. For the Logic gate, tap the **Hysteresis** field and use the assigned multipurpose knob to set the desired hysteresis.
8. For the Logic gate, 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 logic state.
9. Tap anywhere outside the Measurement configuration menu to close it.
10. Tap on a gating-enabled measurement badge to display the vertical gate bars on the waveform for that measurement.
11. To return measurement gating to its default value, open the **Gating** panel and select a gating type of **None**.

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 300).

To set measurement filters you must be taking a measurement. See Add a measurement (on page 80).

Procedure

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 300).

To set measurement limits you must be taking a measurement. See Add Measurements configuration menu overview (on page 170).

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.

Measurement concepts

Measurement variables

By knowing how the instrument makes calculations, you may better understand how to use your instrument and how to interpret your results. The instrument uses a variety of variables in its calculations. These include:

Definition of Base and Top

Base is the value used as the 0% level in measurements such as fall time and rise time.

Top is the value used as the 100% level in measurements such as fall time and rise time. For example, if you set the 10% to 90% rise time, then the instrument calculates 10% and 90% as percentages of Top and Base, with Top representing 100%.

The exact value of Base and Top depends on which Base Top Method you select in the Reference Levels panel of a Measurement configuration menu. It also depends on if you set the reference level to be Global (applies to all measurements set as Global in the Reference Levels panel), or Local (just applies to the measurement that is set to Local).

Base, Top calculation methods

The Base Top calculation method is set in the Reference Levels panel of a Measurement configuration menu.

Auto is the default method, and automatically determines the best Base Top method to use. Most commonly sets the Base Top method to Histogram Mode.

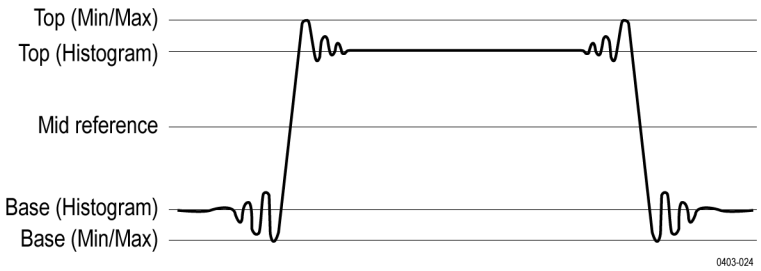
MinMax defines the 0% and the 100% waveform levels as the lowest value and the highest value samples of the waveform record. This setting is best for examining waveforms that have no large, flat portions at a common value, such as sine waves and triangle waves - almost any waveform except for pulses.

The MinMax method calculates the Top and Base values as follows:

Top = Max

and

Base = Min



Histogram Mean uses histogram analysis to select the most common values either above or below the midpoint. Since this statistical approach ignores short-term aberrations (overshoot, ringing, and so on), Histogram is the best setting for examining pulses.

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 (*Record Length* – 1.0) samples unless you are making a gated measurement. When you use cursor gated measurements, it is the location of the right vertical cursor.

Hysteresis Is the hysteresis band of the waveform amplitude.

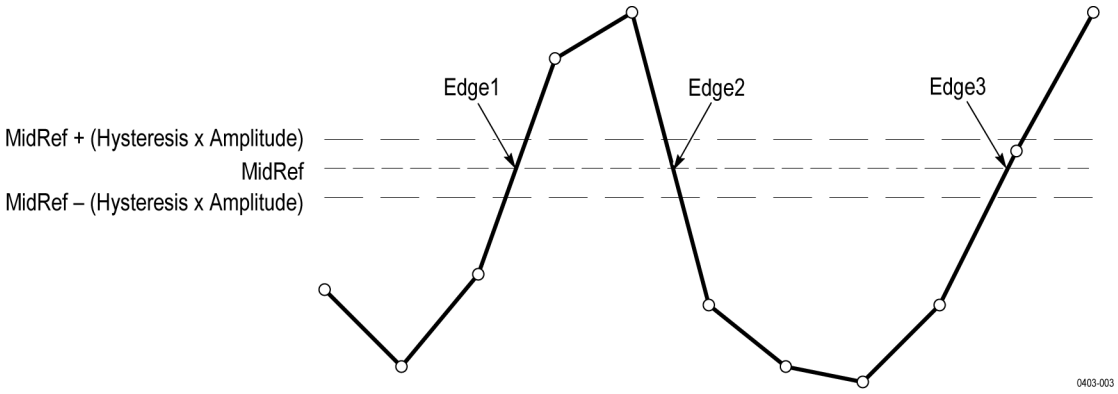
For example, once a crossing has been measured in a negative direction, the waveform data must fall below the hysteresis value of the amplitude from the *Mid* reference point before the measurement system is armed and ready for a positive crossing. Similarly, after a positive *Mid* reference crossing, waveform data must go above the hysteresis value of the amplitude before a negative crossing can be measured. Hysteresis is useful when you are measuring noisy signals, because it allows the oscilloscope to ignore minor fluctuations in the signal.

Edge calculations

Edge1, Edge2, and Edge3 refer to the first, second, and third *Mid* reference edge times, respectively.

An edge can be detected when the waveform is either rising or falling past Midref. The direction of the edges alternates, that is, if Edge1 is rising, Edge2 will be falling.

A rising edge has positive polarity. A falling edge has negative polarity.



The instrument calculates these values as follows:

1. Find the first Mid reference edge in the waveform record or the gated region. This is Edge1.
2. Continuing from Edge1, find the next Mid reference edge in the waveform record (or the gated region) of the opposite direction of Edge1. This is Edge2.
3. Continuing from Edge2, find the next Mid reference edge in the waveform record (or the gated region) of the same direction as Edge1. This is Edge3.

Cycle-cycle measurements are made on each cycle of the waveform. In the diagram above a cycle starts at Edge1 and ends at Edge3.

TPOS is the location of the sample just before the trigger point (the time reference zero sample). In other terms, it contains the domain reference location. This location is where time = 0.

TSOFF is the offset between *TPOS* and the actual trigger point. In other words, it is the trigger sample offset. Values range between 0.0 and 1.0 samples. This value is determined by the instrument when it receives a trigger. The actual zero reference (trigger) location in the measurement record is at (*TPOS* + *TSOFF*).

Missing or out-of-range samples

If some samples in the waveform are missing or off-scale, the measurements will interpolate between known samples to make an appropriate guess as to the sample value. Missing samples at the ends of the measurement record will be assumed to have the value of the nearest known sample.

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.

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.

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.

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
- Filters

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 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.
- When working with reference waveforms, if one or more reference waveforms are used as part of a math waveform, the number of frames in the math waveform is the smallest of all source waveforms (reference, math, or channel waveforms).
- If one or more reference waveforms are used in a math waveform, the record length is equal to the smallest of all the source waveforms (reference, math, or channel waveforms). The math is calculated using the first point from each source, followed by the next point, and so forth. This is true even if the sources have different times between points in the record.

Math waveform editor syntax

You can build math waveforms using the predefined expressions or the equation editor. To help you create valid math waveforms, the following tools will block most illegal entries by disabling any window element that would create an invalid entry in the math waveform expression.

Predefined expressions are accessible using the FFT or Basic Math Types.

The following syntax describes the valid math expressions you can use with the Equation Editor for the Advanced Math Type:

A math expression is composed of settings, functions, scalars and sources.

Settings have the syntax [settingName=settingValue] and are generally applied to measurements. The setting applies to everything to the right of the closing square brackets.

Example: [CoefFileName="highpass_0.25bw.flt"]HighPass(Ch1)

CoefFileName is the setting and is used as the high pass filter on channel 1.

Example: [CoefFileName="highpass_0.25bw.flt"] HighPass(Ch1) + [CoefFileName="lowpass_0.05bw.flt"] LowPass(Ch2)

The high pass filter file is applied to channel 1 and lowpass_0.05bw.flt is applied to channel 2.

Functions, except for basic and logic functions, have the syntax function(source).

In the previous examples the functions are HighPass and LowPass.

Basic and logic functions have the syntax

source1 function source2.

Examples: Ch1 * Ch2

Ch1 AND Ch2

Ch1 >= Ch2

Logic functions, ==|<|>|!=|<=|>=|AND|OR|NAND|NOR|XOR|EQV result in a waveform consisting of binary 0 and 1 values.

Scalars can be integers, floating point values, PI or meas<x>.

Sources can be Ch<x>, Ref<x>, Math<x>

Math waveform differentiation

The math capabilities of the instrument include waveform differentiation. This allows you to display a derivative math waveform that indicates the instantaneous rate of change of the waveform acquired.

Derivative waveforms are used in the measurement of slew rate of amplifiers and in educational applications. You can create a derivative math waveform and then use it as a source for another derivative waveform. The result is the second derivative of the waveform that was first differentiated.

The math waveform, derived from the sampled waveform, is computed based on the following equation:

Yn = (X(n + 1) - Xn) * 1/T

Where: **X** is the source waveform, **Y** is the derivative math waveform, and **T** is the time between samples.

Since the resultant math waveform is a derivative waveform (see the next figure), its vertical scale is in volts/second (its horizontal scale is in seconds). The source signal is differentiated over its entire record length; therefore, the math waveform record length equals that of the source waveform.

Math waveform offset, position, and scale

The settings that you make for offset, scale, and position affect the math waveform you obtain. Here are some tips for obtaining a good display:

- Scale and position the source waveform so that it is contained on the screen. (Off-screen waveforms may be clipped, resulting in errors in the derivative waveform).
- Use vertical position and vertical offset to position your source waveform. The vertical position and offset will not affect your derivative waveform unless you position the source waveform off screen so that it is clipped.

Waveform integration

The math capabilities of the instrument include waveform integration. This allows you to display an integral math waveform that is an integrated version of the acquired waveform.

Use integral waveforms in the following applications:

- Measuring power and energy, such as in switching power supplies.
- Characterizing mechanical transducers, as when integrating the output of an accelerometer to obtain velocity.

The integral math waveform, derived from the sampled waveform, is computed based on the following equation:

$$y(n) = scale \sum_{i=1}^{n} \frac{x(i) + x(i-1)}{2} T$$

Where: **x(i)** is the source waveform, **y(n)** is a point in the integral math waveform, **scale** is the output scale factor, and **T** is the time between samples.

Since the resultant math waveform is an integral waveform, its vertical scale is in volt-seconds (its horizontal scale is in seconds). The source signal is integrated over its entire record length; therefore, the math waveform record length equals that of the source waveform.

Offset and position

When creating integrated math waveforms from live channel waveforms, consider the following:

- You should scale and position the source waveform so that it is contained on screen. (Off screen waveforms may be clipped, which will result in errors in the integral waveform.)
- You can use vertical position and vertical offset to position your source waveform. The vertical position and vertical offset will not affect your integral waveform unless you position the source waveform off screen so that it is clipped.

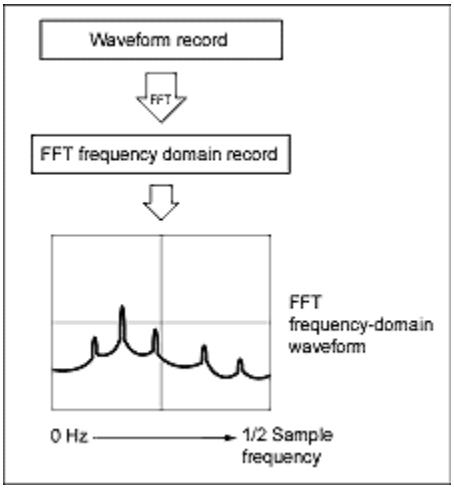
DC offset

The source waveforms that you connect to the instrument often have a DC offset component. The instrument integrates this offset along with the time-varying portions of your waveform. Even a few divisions of offset in the source waveform may be enough to ensure that the integral waveform saturates (clips), especially with long record lengths.

FFT process

The FFT process mathematically converts the standard time-domain signal (repetitive or single-shot acquisition) into its frequency components.

The FFT function processes the waveform record and displays the FFT frequency domain record, which contains the input signal frequency components from DC (0 Hz) to ½ the sample (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.

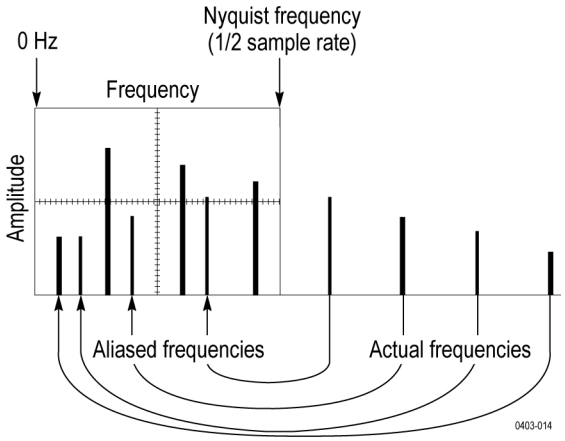
FFT and aliasing

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.



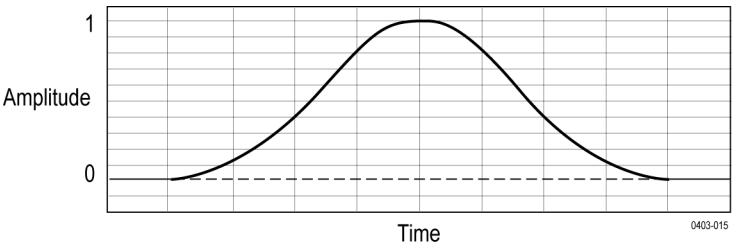
If you have a variable-frequency signal source, another way to observe aliasing is to adjust the frequency slowly while watching the spectral display. If some of the harmonics are aliased, you will see the harmonics decreasing in frequency when they should be increasing or vice versa.

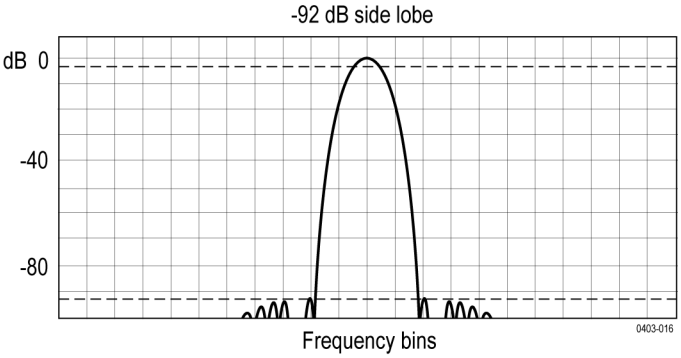
Blackman-Harris FFT window concepts

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 FFT 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

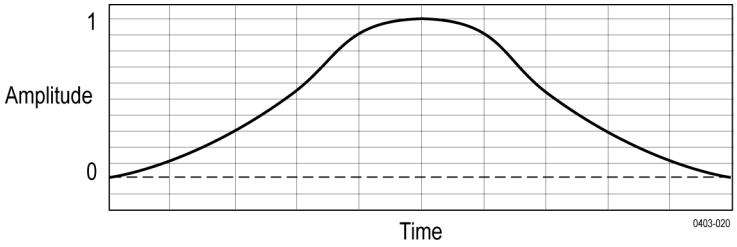
This is the default window function (see the next figure). It is unique in that the time-domain shape of an exponential Gaussian function transforms into a Gaussian exponential shape in the frequency domain.

NOTE: This window provides optimal localization in both the time and the frequency domain.

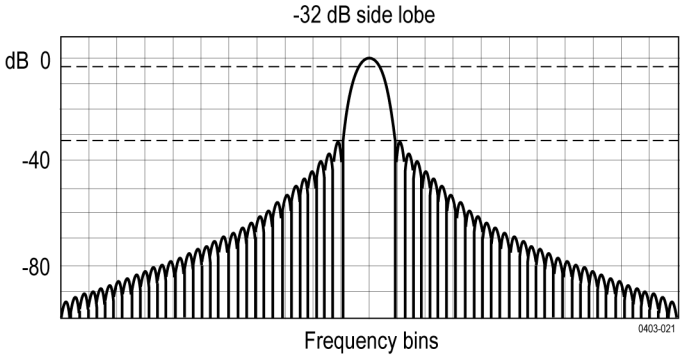
Hanning FFT window

FFT windows have various resolution bandwidths and scallop losses (see the following figure). 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.



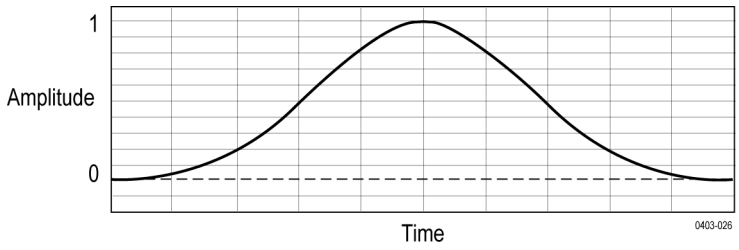
Hanning window side lobe.

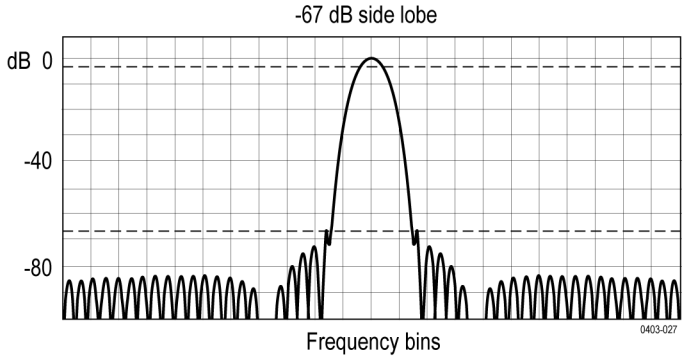
Hamming window

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.

User Defined Filter

User Defined Filter (UDF) can be implemented as Infinite Impulse Response (IIR) or Finite Impulse Response filters (FIR).

The choice of IIR or FIR filter is based on the design requirements and application. For example, when linearity of phase is important, a FIR filter is a better choice. Normally, IIR filters have asymmetric impulse responses and non-linear phases.

Following are the list of supported filter types:

- Low-Pass
- High-Pass
- Band-Pass
- Band-Stop

- All-Pass
- Hilbert
- Differentiator
- Raised Cosine
- Root-Raised Cosine

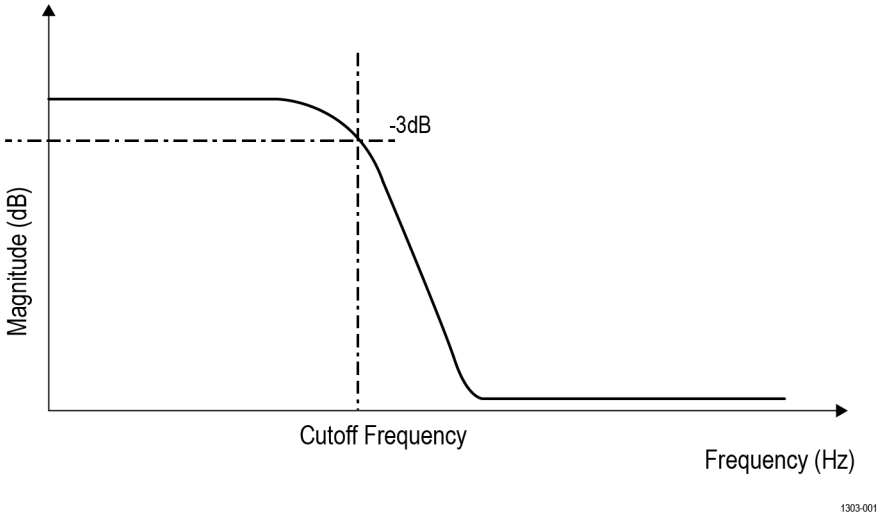
Following are the list of supported filter responses:

- Butterworth
- Chebyshev I
- Chebyshev II
- Elliptical
- Gaussian
- Bessel-Thompson
- Custom

The filters are designed up to [Sample Rate (Fs)]/2, hence the UDF magnitude plot shows the frequency axis from 0 Hz to Fs/2 Hz, where Fs is the Sample Rate Frequency. At the cutoff frequency, the gain will be 0.707 (or -3dB). For example, if passband gain is 0 dB then the gain at cutoff frequency should be -3 dB.

Low-Pass Filter (LPF)

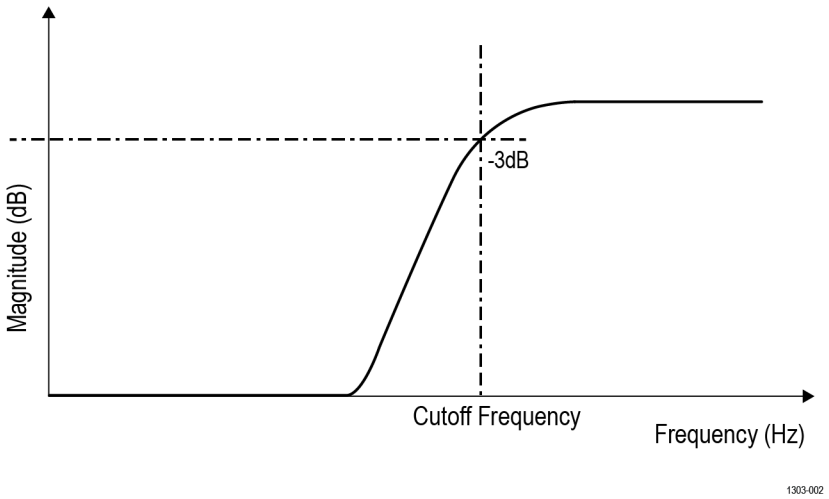
A low-pass filter passes all the frequencies up to the specified cutoff frequency. It rejects all the frequencies above the cutoff.



LPF are used to remove background and high frequency noise.

High-Pass Filter (HPF)

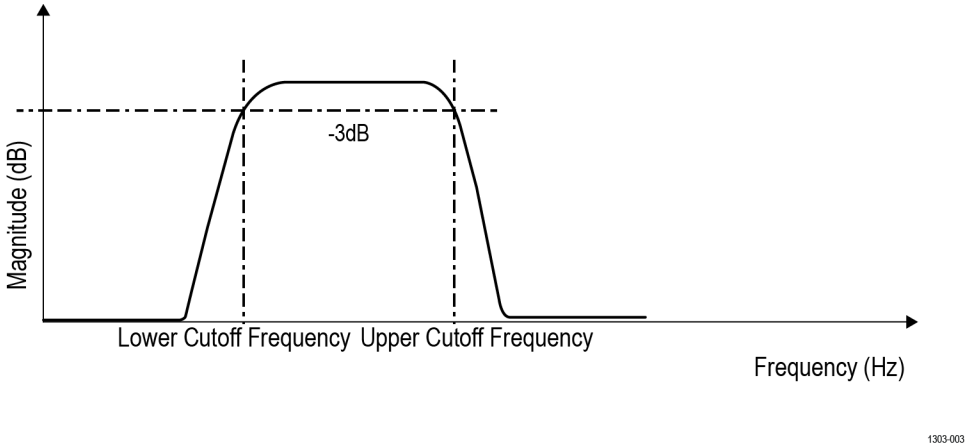
A high-pass filter passes all the frequencies above the specified cutoff frequency. It rejects all the frequencies below the cutoff.



HPF can be used to remove DC and low-frequency components. Both LPF and HPF are commonly used in HSS (High Speed Serial) and data communication applications.

Band-Pass Filter (BPF)

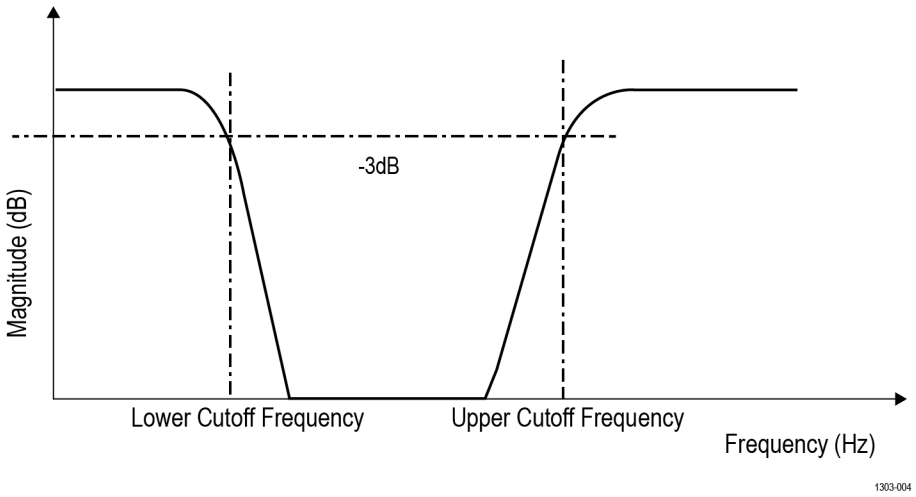
A band-pass filter passes the band of frequencies between configured lower and upper cut off frequencies. Specify the upper and lower cut off frequencies for the filter.



BPF is used in HSS (High Speed Serial) analysis for doing jitter measurements over a frequency range of interest. BPF can be used in Pre-amplification, Equalization and Control in Audio Systems applications.

Band-Stop Filter (BSF)

A band-stop filter passes all frequencies except a configured frequency band between lower and upper cut off frequencies.

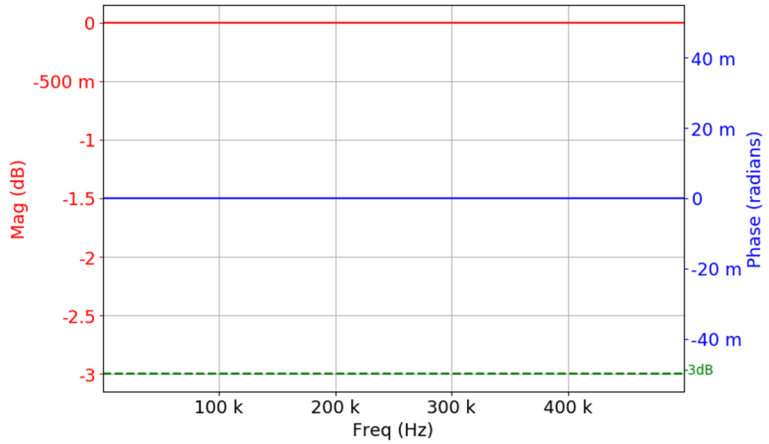


BSF filters are extensively used in Medical electronic systems.

All-Pass Filter

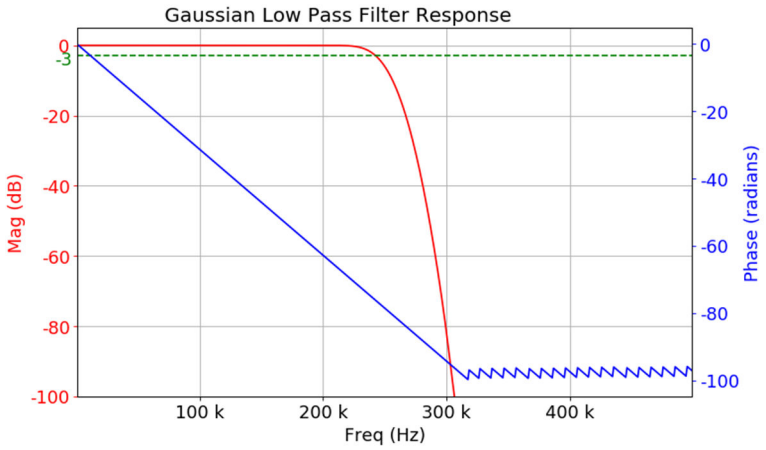
The All-pass filter amplitude is constant and passes all frequencies between DC to Fs /2. Enter the delay in seconds, which introduces appropriate delay on the filtered output MATH waveform, which is equivalent to phase shift in the frequency domain. The application computes and displays the plots of magnitude and phase responses.

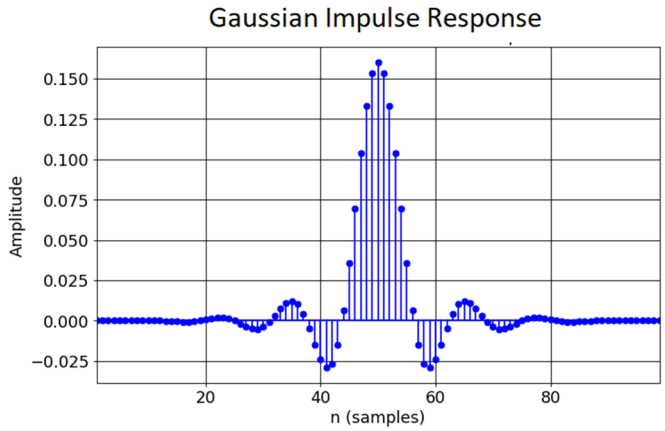
NOTE: Displays magnitude and Phase response plot for all-pass filters.

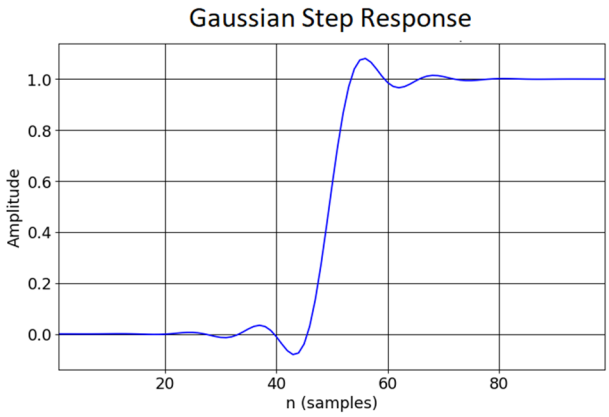


Gaussian

This is an FIR filter designed using the Gaussian window method. This function computes the coefficients of a finite impulse response filter. The different types of filters which can be designed using the Gaussian window method are Low-pass, High-pass, Band-pass, and Band-stop.





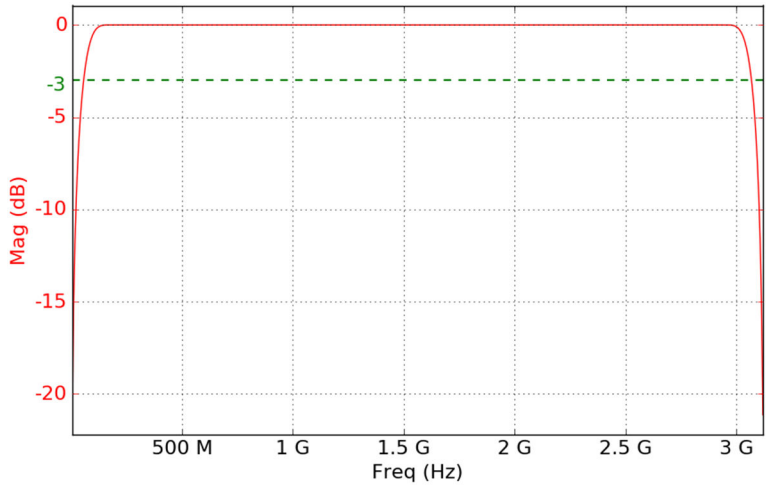


The standard deviation is a user input that defines the Gaussian window width.

Hilbert

The ideal Hilbert transform filter has a gain of 1 dB at all frequencies and shifts the phase of all frequencies by 90 degrees. Hilbert filter can be used to create quadrature signals over a wide frequency range.

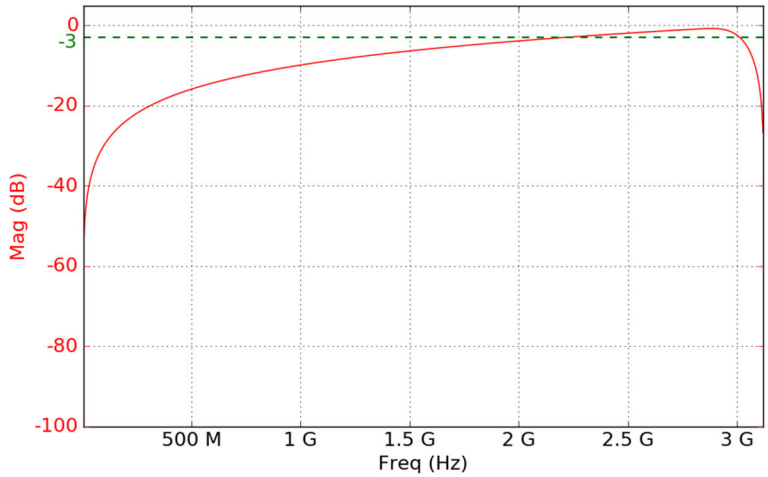
NOTE: Displays magnitude response plot for Hilbert filters.



Differentiator

The ideal differentiator is a high pass filter that shifts phase by 90 degrees. Its frequency response would be linear all the way from DC to Fs/2.

NOTE: Displays magnitude response plot for Differentiator filters.

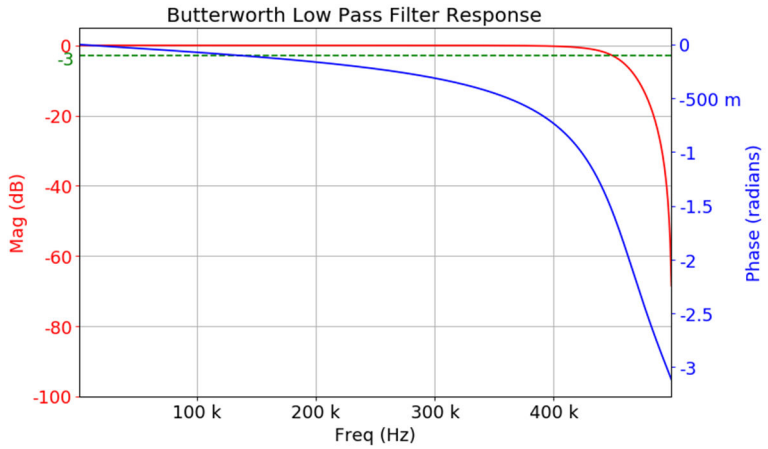


If the REF waveform sample rate does not match with Scope horizontal sample rate, then the user must change the oscilloscope sample rate to match with the REF sample rate.

Butterworth

Butterworth filters do not have passband and stop-band ripple, and commonly known as maximally flat magnitude filters. This has the flattest amplitude response of all the available filters.

The plot diagram is the example showing a low-pass filter. This filter also supports High-pass, Band-pass, and Band-stop filters. The application computes and displays plots of magnitude, phase, impulse, and step responses.



As the filter order is increased, the passband and stop-band performance improve, with the transition from passband to stop-band becoming sharper. The Butterworth method is best suited for design problems that focus on controlling passband and stop-band ripple.

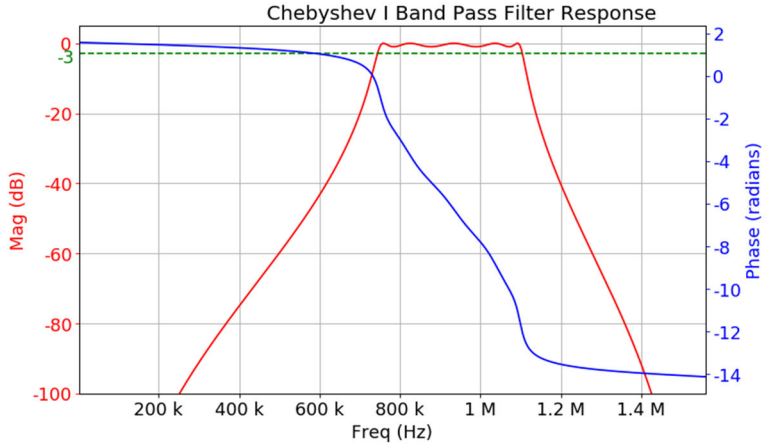
This filter is used in electrophysiology and also a good choice for DC measurement applications.

Chebyshev I

There are two types of Chebyshev filter namely Type-1 and Type-2. Generally, Type 1 is referred to as a regular filter, and this is the most common Chebyshev filter. The transition from passband to stop-band is more rapid than the Butterworth filter.

Type-1 filter has an equi-ripple in passband and no-ripple in the stop-band. The application provides control of passband ripple as part of this filter.

The plot diagram is the example showing for low pass filter, this is also applicable for High-pass, Band-pass, and Band reject filters.

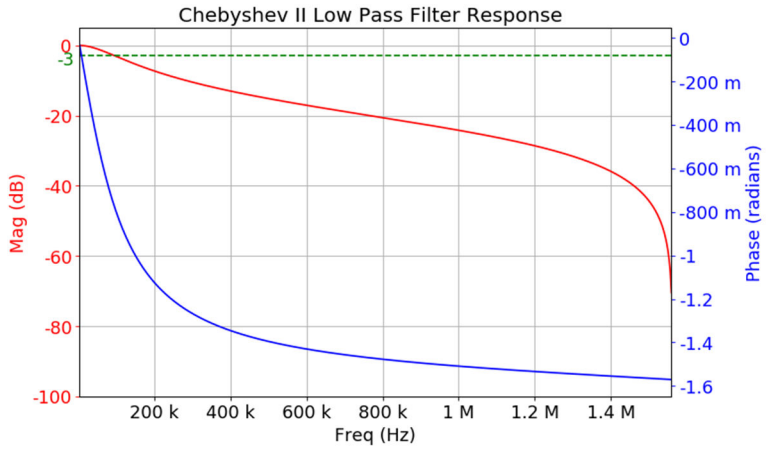


The application computes and displays plots of magnitude, phase, impulse, and step responses.

Chebyshev II

Chebyshev II is also known as the inverse Chebyshev filter. The type-2 has equiripple in stop-band and no-ripples in the passband. These filters have a maximally flat passband response. The application provides control of Stop-band attenuation as part of this filter.

The plot diagram is the example showing for low-pass filter, this is also applicable for High-pass, Band-pass, and Band reject filters.



The application computes and displays plots of magnitude, phase, impulse, and step responses.

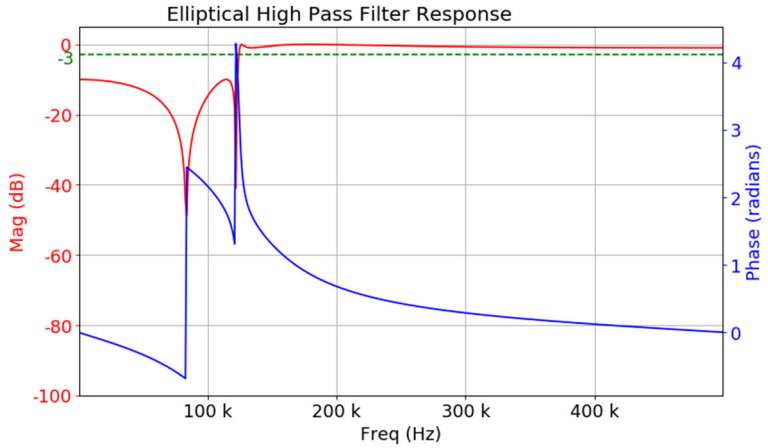
NOTE: Chebyshev II filter considers stop-band attenuation value to design the filter. The passband gain is measured above the stop-band attenuation.

Elliptical

Elliptical filters have passband and stop-band ripple and the sharpest roll-off near the cutoff frequency. This type of filter contains ripple behavior in both the passband and stop-band.

The elliptical filter has fastest roll-off and lowest order filter than others. The application provides control of Passband ripple and Stop-band attenuation as part of this filter.

The plot diagram is the example showing high-pass filter. This filter also supports Low-pass, Band-pass and Band reject filters. The application computes and displays plots of magnitude, phase, impulse, and step responses.



This filter is used in many RF and microwave applications where a very fast transition between the passband and stop-band frequencies is required.

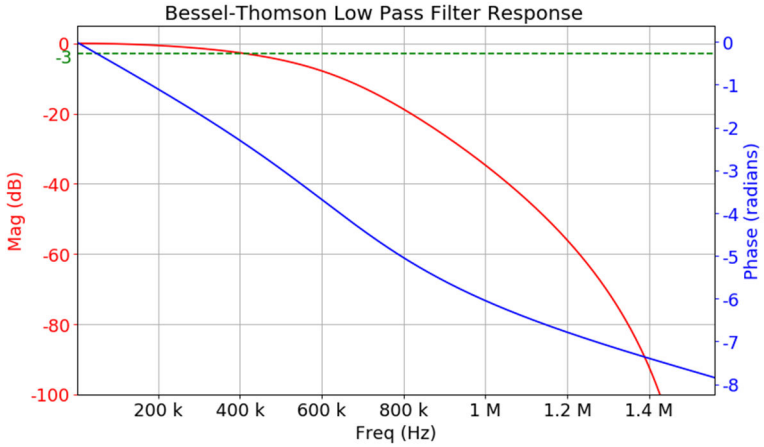
Bessel Thompson

The Bessel filter is optimized to obtain a better transient response due to a linear phase that is a constant delay in the passband.

The plot diagram is the example showing a low-pass filter. This filter supports High-pass, Band-pass and Band reject filters. The application computes and displays the plots of magnitude, phase, impulse, and step responses.

The Bessel Thompson class of filters are commonly used as the analog bandwidth limit filter in oscilloscope channels. It is also used as the filter response type for optical reference receiver mask testing applications. The reason for its popularity is that it is the closest one can get to realizing an ideal Gaussian filter response that is not physically realizable.

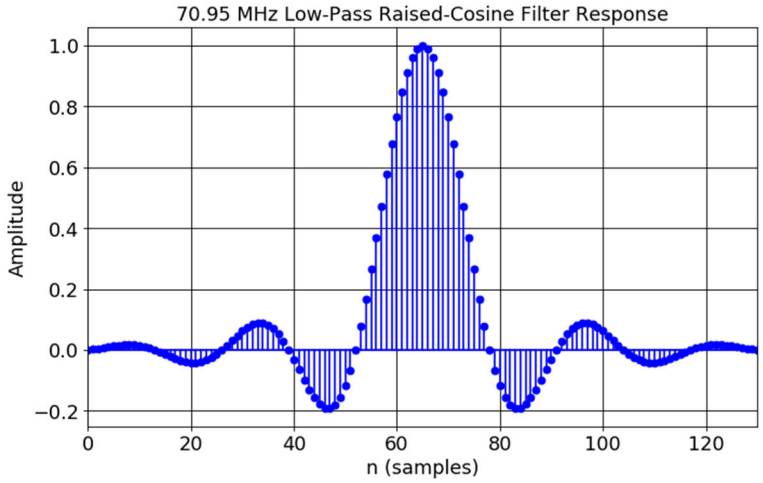
Bessel Thompson can be used in some RF and in particular audio applications where the preservation of the wave shape and phase of components within a signal are important.

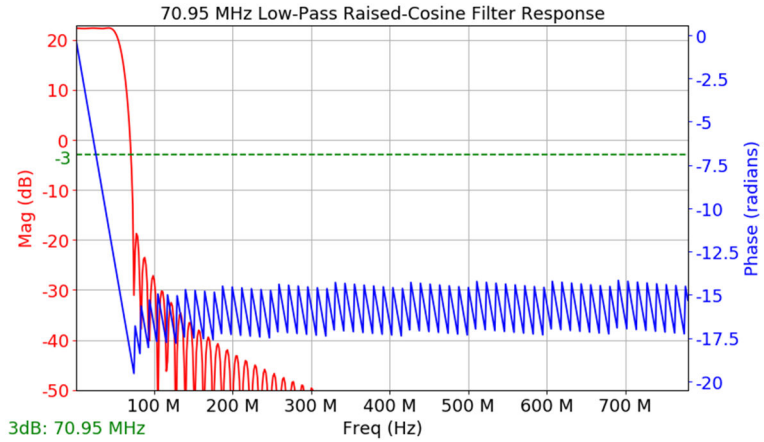


These filters are used in applications where a constant group delay is critical, such as in analog video signal processing.

Raised Cosine (RC) and Root Raised Cosine (RRC) filters

Raised Cosine and RRC filters are low-pass filters.





Ideal pulse shaping filters will have infinite number of taps. The UDF will window the filter span based on number of symbols. Symbols configuration helps in setting window width. RC and RRC need number of samples per symbol, hence Symbol Duration configuration is utilized. Samples per symbol is the product of current sample rate of oscilloscope and symbol duration. Since the sample rate is inherently a configuration of filter, the filter response changes for different sample rate. The UDF designs just for the current oscilloscope sample rate.

The filter order is computed as `Symbols x Symbol Duration x Sample Rate`.

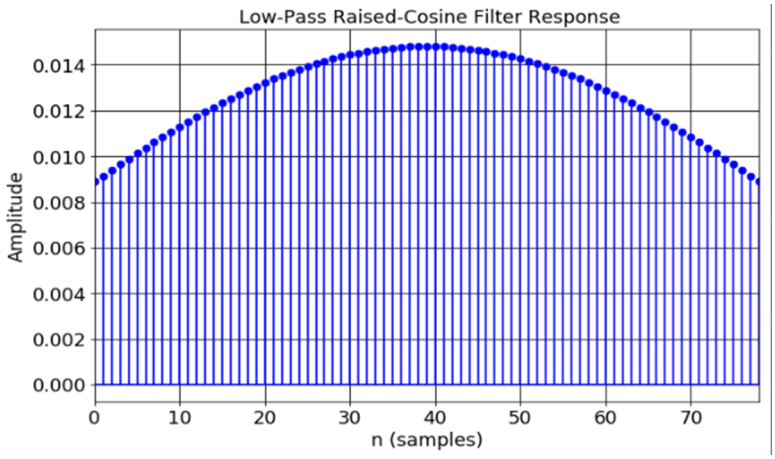
Number of taps is an odd number, in order to maintain even symmetry in impulse response.

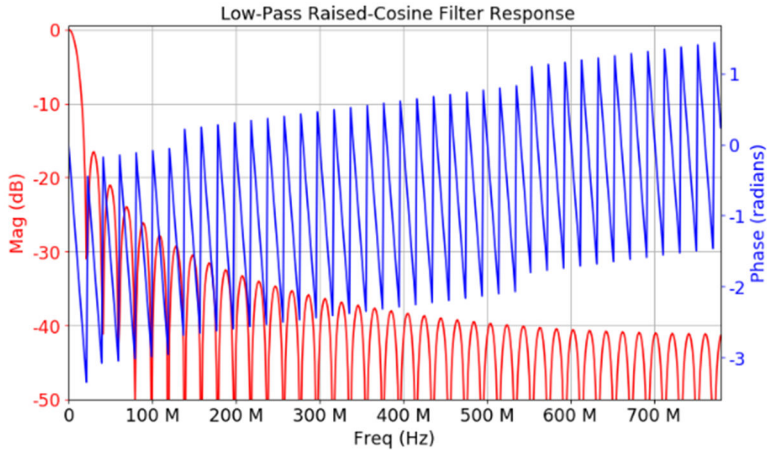
Troubleshooting RC and RRC filters

For a fixed sample rate, rate of transition falls and side lobes gain can be varied by changing Symbols (filter span). Impulse response is a good indicator of transition.

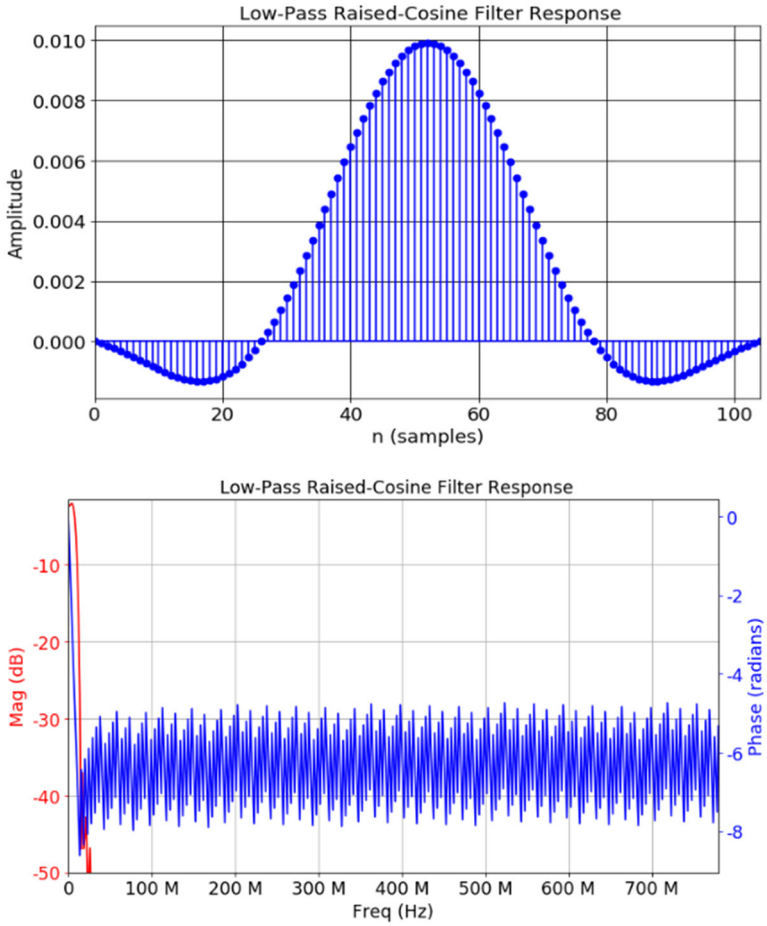
Example: RC filter with Symbol Duration = 50 ns, Roll off factor = 0.5, and Sample rate = 1.5625 GS/s

For Symbols = 1:





For Symbols = 4:



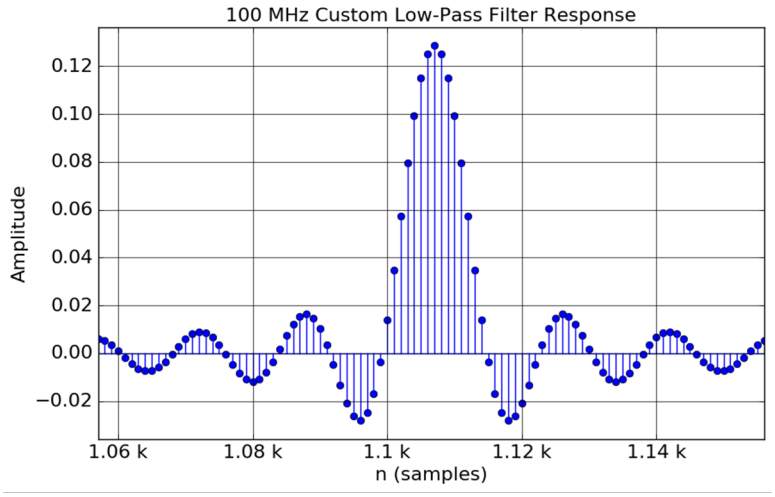
As impulse response starts, looking closer to a sinc response, the rate of magnitude transition increases rapidly to an extent where gain of output waveform start decreasing and also ripples in output waveform increases. It is advised to decrease the sample rate and symbols configuration in such situations. There will be a constant group delay in filtered output which can be removed by UDF “All Pass filter” if necessary.

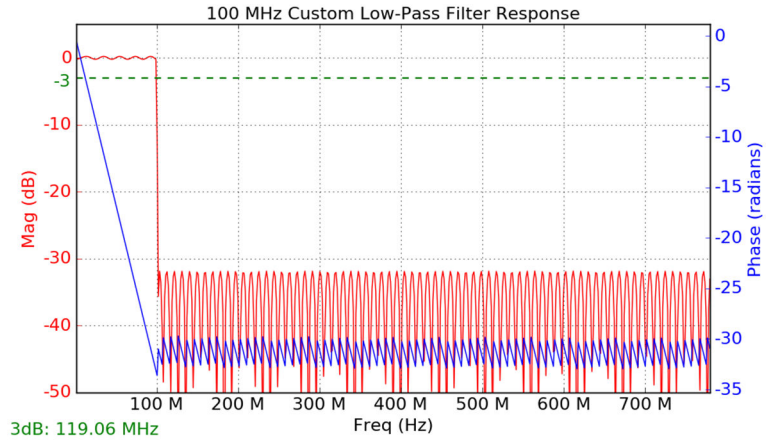
Custom filters

The custom filter order is calculated using the remez order formula. It depends on the transition width, pass band ripple, stop band attenuation, and the sample rate. The maximum supported filter order is 15k. So, the transition width and other parameters can be adjusted accordingly to get the filter order below 15k.

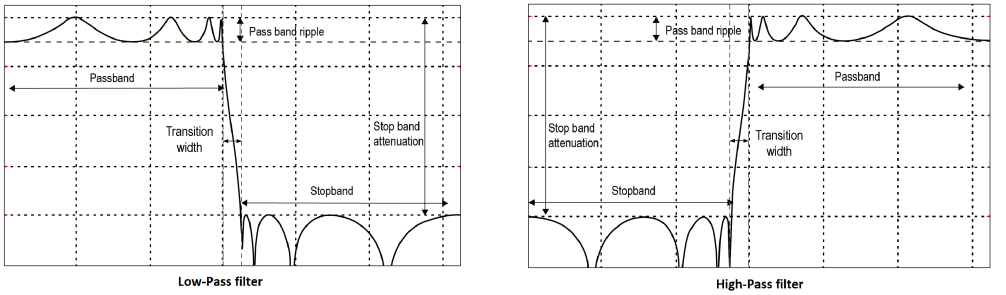
Constraints on transition width:

- In case of low-pass filter the maximum value of transition width allowed is (0.45 * sample rate) - cutoff frequency.
- In case of high-pass filter it cannot exceed the cutoff frequency.
- In case of band-pass or band-stop filters the maximum value of transition width can be the minimum of the low cutoff frequency and (0.5 * sample rate - high cutoff frequency).





Key filter parameters



- A filter's response to different frequencies is characterized as passband, transition band, or stop-band.
- Passband response is the filter's effect on frequency components that are passed through (mostly) unchanged.
- Stop-band response: Band of frequencies that are highly attenuated post filtering.
- Transition band (Transition width): It represents frequencies in the middle, which may receive some attenuation but are not removed completely from the output signal.
- Cut off Frequency: Used to specify the bandwidth of high-pass, low-pass, band-pass, and band-stop filters. The cutoff frequency is the point at which the magnitude is close to -3 dB from passband gain. This represents a 3 dB point.
- Filter Roll off rate: The slope of the transition band of a filter is referred to as its roll off rate. This is commonly specified in dB per octave or in dB per decade.
- Filter order: The order refers to the order of the filter's system function. The filter roll-off is a function of the filter order.

- The transition region: This region is between passband and stop-band enclosing the cutoff frequency and is defined as the transition band. Larger the filter roll off rate smaller the transition region.
- Impulse response: The impulse describes the transfer function of a filter in the time domain. The effect of a filter on the signal input resulting in a filtered output. Therefore, it is essential to understand the filter responses for a good filter design. This is the output from the filter when the input is unit impulse.
- Step response: This response represents the integral of the impulse response for the digital filter transfer function. This is the output from the filter when the input is a unit step function.
- Stop-band attenuation: Describes dB value of stop-band attenuation down from the peak passband value.
- Passband Ripple: Describes dB of peak-to-peak passband ripple in the passband region.

NOTE: Filter Designer may not be able to achieve the desired response using different design methods, regardless of the filter order. So, you are expected to tune the filter based on the filter response and any warning messages.

- You can design a filter for the configured cutoff frequency and the corresponding filter response is shown. However, the filtered signal on math is shown when the cutoff frequency is between 0.05 to 0.45 times the sample rate. You need to adjust the oscilloscope sample rate to one of the valid sample rates for which the filter is designed to see the filtered signal.
- Filter Blanking time: Filter blanking time is the duration of the filter output that is suppressed. The blanked portion of the output is not included in the measurement statistics or in any plots. To overcome this, increase the input waveform to a higher record length if MATH does not show the filtered waveform. This happens due to filter blanking. If there is no output on the math waveform, it also could be due to this reason.
- The magnitude response shows the Cutoff frequency annotation at bottom left of the graph. This is obtained by finding intersection of magnitude curve with -3 dB horizontal line. This is applicable for all filters except All Pass filter.

Known limitations of the user-defined filter

Following are the limitations to design user defined filter:

- The delay range is (-5 * configured horizontal scale) to (+ 5 * configured horizontal scale). The MATH supports only positive values of delay. So user inputs are in the range of 0 to (5 * configured horizontal scale).
- Response and output magnitude will mismatch in FIR Filters. This applies to All-Pass, Hilbert, and Differentiator.

Filter file format

Filter file is unique to Tektronix User Defined Filter software. When filter file is saved, it will be in the below format.

```
// Do not edit. Generated from Tektronix Filter Editor.
// FIR or IIR Filter
// Filter Header
// Sample Rate; Filter Coefficients
```

NOTE: Filter coefficients are saved for all the valid sample rates for which filter can be designed.

For Example:

1. Filter file created for 2nd order low pass Butterworth filter.

```
// Do not edit. Generated from Tektronix Filter Editor.
// Tool converts IIR filter response to FIR filter response.
// {filter_type: lowPass, filter_name: butterworth, filter_order: 2, fs: 6250000000.0, fir_taps:
 5001, cutoff_freqs: 1000000000.0}
6250000000.0;1.453238838770423980e-01,3.881643214198569058e-01, and so on
```

2. Filter file created for Hilbert filter.

```
// Do not edit. Generated from Tektronix Filter Editor.
// FIR filter.
// {filter_type: hilbert}
@6250000000.0;-7.035076590000000241e-17,1.304761389999999890e-04, and so on
```

NOTE: If the recalled filter file is not in the above format, it is unable to restore the filter settings. However, filtered signal will display on math if the sample rate in the filter file matches with the oscilloscope sample rate.

Measurement algorithms

Amplitude measurement algorithms

AC RMS measurement algorithm

AC RMS is the true Root Mean Square of the data points about the Mean (μ). This measurement can be made across the entire record or on each cycle in the record.

Area measurement algorithm

Area is the arithmetic area for one waveform. The area measured above ground is positive. The area measured below ground is negative. Remember that one waveform is not necessarily equal to one cycle. For cyclical data you may prefer to use the cycle area rather than the arithmetic area.

$$Area = \int_{Start}^{End} Waveform(t)\,\mathrm{dt}$$

0020-012

.

Amplitude measurement algorithm

Amplitude is the difference between the Top value and the Base value.

Amplitude = Top - Base

Base measurement algorithm

Base is calculated using the selected Base Top method. Base is the most common data value below the midpoint of the waveform, when the default Base Top method Histogram Mode is selected. This measurement can be made across the entire record or on each cycle in the record.

Integration algorithm

The integration algorithm used by the instrument is as follows:

$$\int_A^B W(t)dt \quad \text{is approximated by} \quad \int_A^B \hat{W}(t)dt \quad \text{where:}$$

0020-025

W(t) is the sampled waveform

$\hat{W}(t)$ 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,

$$\int_A^B (W(t))^2 dt \text{ is approximated by } \int_A^B \left(\hat{W}(t)\right)^2 dt \text{ where:}$$

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:

$$\int_A^B \left(\hat{W}(t)\right)^2 dt = s \times \sum_{i=A}^{B-1} \frac{(W(i))^2 + W(i) \times W(i+1) + (W(i+1))^2}{3}$$

0020-028

where *s* is the sample interval.

Maximum measurement algorithm

Maximum is the maximum data point. Typically the most positive peak voltage.

Mean measurement algorithm

Mean is the arithmetic mean of the data points. Remember that one waveform is not necessarily equal to one cycle. For cyclical data you may prefer to use the cycle mean rather than the arithmetic mean.

$$Mean = \frac{\int_{Start}^{End} Waveform(t)\mathrm{dt}}{(End - Start) \times SampleInterval}$$

0020-017

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. Overshoot values should never be negative (unless Top or Base are set out-of-range).

$$\text{Negative Overshoot} = \frac{\text{Base} - \text{Min}}{\text{Amplitude}} \text{X} 100\,\%$$

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. This value should never be negative.

$$\text{Positive Overshoot} = \frac{\text{Max} - \text{Top}}{\text{Amplitude}} \text{X} 100\,\%$$

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.

$$RMS = \sqrt{\frac{(data1^2 + data2^2 ...)}{(end - start)}}$$

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.

Eye measurement algorithms

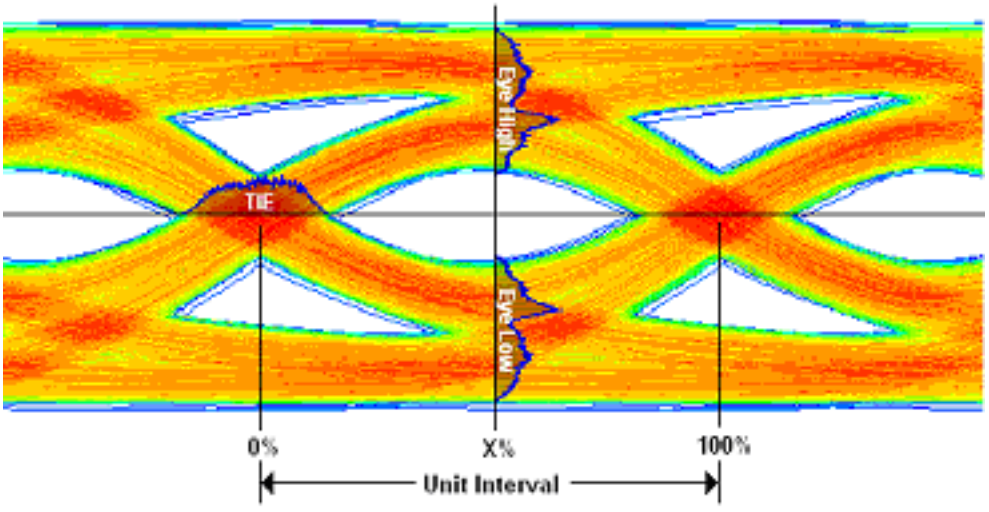
Eye Height measurement algorithm

If analysis method is Parametric, Eye Height is the minimum vertical eye opening at the mid of the unit interval. This measurement is made across the entire record.

If analysis method is Histogram, Eye Height is calculated non-parametrically based on the rendered eye diagram. The eye height measurement can be controlled by configuring the parameters.

Eye high

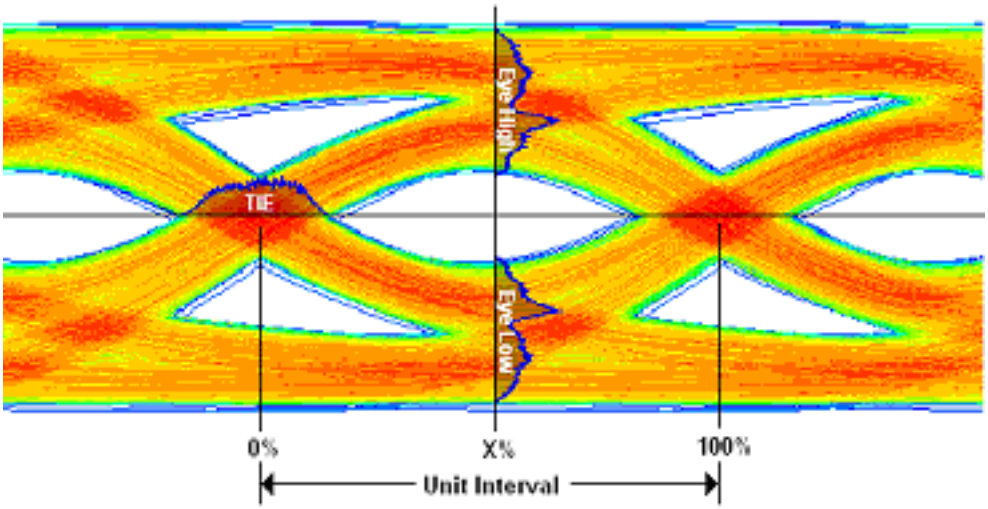
Eye High calculates the voltage at a selected horizontal position across the unit interval, for all High bits in the waveform. You specify the offset at which the measurement takes place from 0% to 100% of the unit interval. Configure the measurement to include all bits, only transition bits, or only 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

If analysis method is Parametric, Eye Width is the minimum horizontal eye opening at the user-specified reference level. This measurement is made across the entire record.

If analysis method is Histogram, Eye Width is calculated non-parametrically based on the rendered eye diagram. The eye width measurement can be controlled by parameters.

Height@BER

Height@BER is the Eye Height at a specified Bit Error Rate (BER). This extrapolated value predicts a vertical eye opening that will be violated with a probability equal to the BER. It is generally not equal to the eye height actually observed in any given acquisition. A single Height@BER value, in the given interval, is determined for each acquisition by means of Q-scale extrapolation.

Q-factor

Quality Factor is the ratio of eye size to noise.

The final measurement value would be computed according to the equation below:

```
Q-factor = [mean(EyeHigh) - mean(EyeLow)] / [stddev(EyeHigh) + stddev(EyeLow)]
```

Where:

Eye High: the sample values of positive UI at x%.

Eye Low: the sample values of negative UI at x%.

For more details refer Eye Height

Width@BER

Width@BER is the Eye Width at a specified Bit Error Rate (BER). This extrapolated value predicts a horizontal eye opening that will be violated with a probability equal to the BER. It is generally not equal to the eye width actually observed in any given acquisition. A single Width@BER value is determined for each acquisition, by means of RJ-DJ separation analysis.

Jitter measurement algorithms

AC Common Mode

AC Common Mode (Pk-Pk) is the peak-to-peak amplitude of the common mode of the sources. Voltage sources are typically filtered to include only the frequency components above the cutoff frequency (30 kHz). The filter can be disabled in the measurement configuration. The measurement can be configured to take one or two sources, and the cutoff frequency may be enabled or disabled. The application calculates this measurement using the following equation:

Peak-to-Peak (High Pass Filter((Source1 + Source2)/2))

Bit Amplitude measurement algorithm

Bit Amplitude is the difference between the levels of the "1" and "0" bits surrounding each transition, measured over a specified range at the center of the recovered unit interval. This measurement is made on each transition bit (Mean) or across the entire record (Mode).

Bit High measurement algorithm

Bit High is the amplitude of a "1" bit. The amplitude is measured over a user-specified portion at the center of the recovered unit interval. This measurement is made on each high bit in the record (Mean) or across the entire record (Mode).

Bit Low measurement algorithm

Bit Low is the amplitude of a "0" bit. The amplitude is measured over a user-specified portion at the center of the recovered unit interval. This measurement is made on each low bit in the record (Mean) or across the entire record (Mode).

DC Common Mode measurement algorithm

DC Common Mode is the arithmetic mean of the common mode of two sources. This measurement is made across the entire record.

Differential Crossover measurement algorithm

Differential Crossover is the voltage level of a differential signal pair at the crossover point(s). This measurement is made at each crossover point in the record.

SSC Freq Dev measurement algorithm

SSC Freq Dev is the spread spectrum clock frequency deviation from the nominal frequency in ppm (parts per million).

Clock recovery is used on the measurement. The method is constant clock mean.

SSC Modulation Rate measurement algorithm

SSC Modulating Rate is the modulating frequency of a spread spectrum clock. It is the rate that the clock frequency changes.

Clock recovery is used on the measurement. The method is constant clock mean.

TIE

TIE (Time Interval Error) is the difference in time between an edge in the source waveform and the corresponding edge in a reference clock. The reference clock is usually determined by a clock recovery process performed on the source waveform. For Explicit-Clock clock recovery, the process is performed on an explicitly identified source.

If the Signal Type is Clock

The application calculates Clock TIE measurement using the following equation:

$$TIE_n^{Clock} = T_n^{Clock} - T'_n{}^{Clock}$$

Where:

TIEClock is the clock time interval error.

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.

T'Data is the corresponding edge time for the specified reference clock.

The subscript k is used to indicate that there is one measurement per actual edge.

T/nT Ratio measurement algorithm

T/nT Ratio is the ratio in dB of the midpoint level of the 1st bit after each transition to the midpoints of the 2nd and subsequent non-transition bit levels prior to the next transition. This measurement is made for each non-transition bit in the record.

DCD

Duty Cycle Distortion (DCD) is the peak-to-peak amplitude for that portion of the deterministic jitter directly correlated with signal polarity, that is the difference between the mean positive edge displacement versus that on negative edges. A single DCD value is determined for each acquisition, by means of RJ-DJ separation analysis.

DDJ

Data-Dependent Jitter (DDJ) is the peak-to-peak amplitude for that portion of the deterministic jitter directly correlated with the data pattern in the waveform. It includes the effects of both ISI and Duty Cycle Distortion (DCD). A single DDJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

There are two ways to compute DDJ. They are **Time Domain** and **Spectral Domain**. The method can be set in the User preference menu. This include the effects of both ISI and Duty Cycle Jitter (DCJ).

DJ

Deterministic Jitter (DJ) is the peak-to-peak amplitude for all timing errors that follow deterministic behavior. A single DJ value is determined for each acquisition, by means of RJ-DJ separation analysis.

Dual Dirac deterministic jitter

Dual Dirac Deterministic Jitter (DJ–δδ) the peak-to-peak magnitude for all timing errors exhibiting deterministic behavior, calculated based on a simplifying assumption that the histogram of all deterministic jitter can be modeled as a pair of equal magnitude dirac functions (impulses). A single DJ–δδ value is determined for each acquisition, by means of RJ-DJ separation analysis.

F/2 measurement algorithm

F/2 is the peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by 2. This measurement is made across the entire record.

F/2 is reported as non-zero even if it coincide spectrally with DCD.

F/4 measurement algorithm

F/4 is the peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by 4. This measurement is made across the entire record.

F/4 is reported as non-zero even if it coincide spectrally with DDJ.

F/8 measurement algorithm

F/8 is the peak-to-peak amplitude of the periodic jitter occurring at a rate of Fb (data rate) divided by 8. This measurement is made across the entire record.

F/8 is reported as non-zero even if it coincide spectrally with DDJ.

J2

J2 is Total Jitter at a Bit Error Rate (BER) value of 2.5E-3. This statistical value predicts a peak-to-peak jitter that will only be exceeded with a probability equal to the BER.

J9

J9 is Total Jitter at a Bit Error Rate (BER) value of 2.5E-10. This statistical value predicts a peak-to-peak jitter that will only be exceeded with a probability equal to the BER.

CC-Jitter (Cycle to Cycle Jitter)

CC-Jitter measures the clock period change between any two adjacent cycles. Cycle-to-Cycle jitter is measured by applying a first order difference operation to the period jitter.

Jitter Summary measurement

The Jitter Summary measurement is a predefined set of jitter measurements displayed in a single badge. The measurements include Data Rate, Pattern Length, TIE, TJ@BER, RJ, DJ, DJ-δδ, PJ, DDJ, ISI, DCD, Eye Width@BER, and RJ-δδ. It also includes Eye Height@BER when Noise Analysis is enabled. This measurement also adds Eye Diagram, Composite Jitter Histogram, Tie Spectrum, and Bathtub plots to the screen.

NPJ

Non-Periodic Jitter (NPJ) is the dual-dirac magnitude of that portion of Bounded Uncorrelated Jitter (BUJ) that is not periodic. Since it is not periodic and is not correlated with the data pattern, NPJ is frequently difficult to distinguish from (Gaussian) RJ.

This component of jitter is not analyzed by default, but you can enable it by switching the jitter analysis mode to Spectral + BUJ. Since it typically requires high populations to distinguish, you may need to acquire multiple waveforms before jitter results are available when Spectral + BUJ mode is enabled.

Phase noise

The Phase Noise measurement performs a jitter measurement, converts the result into the frequency domain, and reports the rms jitter integrated between two specific frequencies selected by the user.

A 10^{th} decade averaging filter is applied on the phase noise spectrum to smooth the plot. Smoothing option is on by default. You can turn off the smoothing filter from the measurement configuration panel. For the phase noise calculation, smoothed Rj spectrum is used when **Smoothing** option is enabled, else original Rj spectrum is used.

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.

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.

SJ@Freq measurement algorithm

SJ@Freq refers to sinusoidal jitter occurring at or near a frequency. It is defined as the peak-to-peak amplitude of the sinusoidal components of deterministic jitter within a designated frequency band around the target frequency.

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 δδ 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. 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.

Transfer Function algorithm

A Transfer Function plot displays the magnitude ratio of the frequency spectra of two measurements on logarithmic axes. This plot provides a useful way to depict a system's response to stimuli at different frequencies and to identify poles and zeros in a system's characteristic equation.

Suppose that x(t) represents a jitter measurement at the input of a device and y(t) represents the corresponding jitter measurement at the output. The Transfer Function plot can be used to represent the following function, where X(f) is the Fourier transform of x(t):

H(f) = Y(f) / X(f)

The horizontal axis of the Transfer Function plot extends to the Nyquist frequency of X or Y, whichever is lower.

Jitter Analysis Method

Breakdown of jitter (Jitter map)

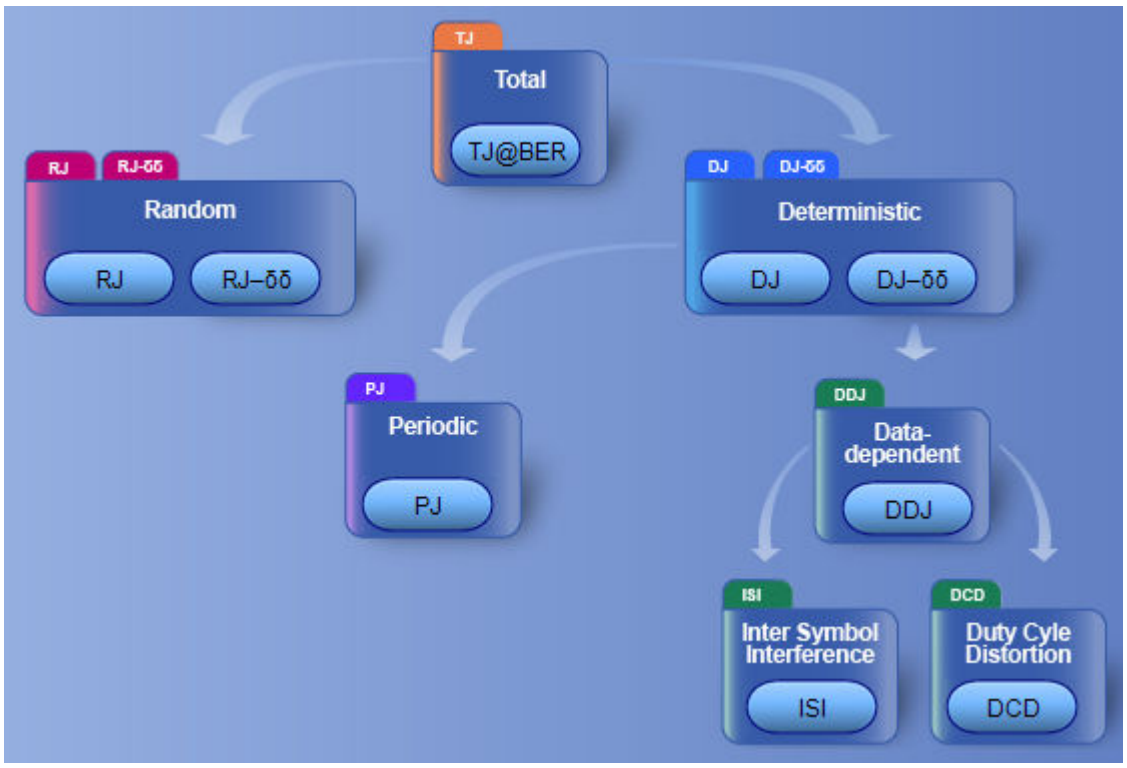
Jitter is defined as the short-term variations of a digital signal's significant instants from their positions in time. The breakdown of jitter into components such as RJ, PJ, and DDJ uses a model-based approach. This involves proposing a mathematical model for the overall jitter that comprises various jitter components. These components are distinguishable based on observable characteristics, and their combination to form the overall jitter distribution follows well-established mathematical principles.The model is hierarchical, and is represented by a jitter map.

Depending on the selected Jitter Separation Model, two distinct jitter maps are utilized. To select the Jitter Seperation Model, follow the steps:

1. Tap **Utility > User Preferences > Jitter and Eye Analysis**.
2. Select the **Jitter Separation Model** as **Spectral Only** or **Spectral + BUJ**.

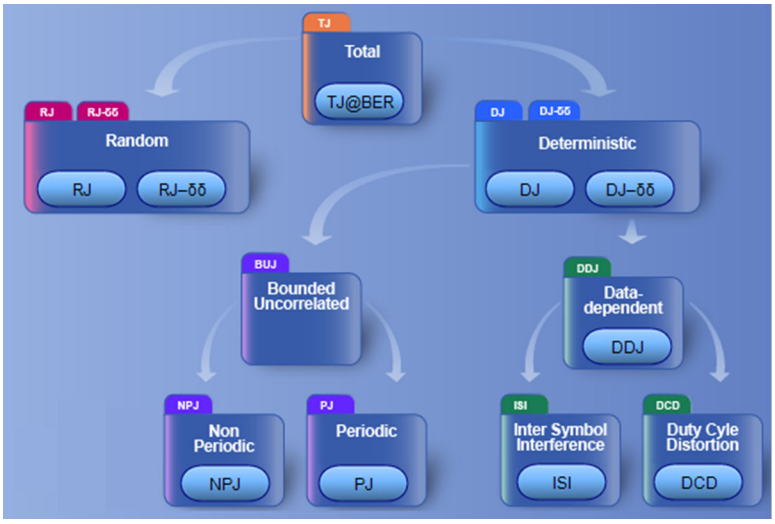
The two different jitter maps are:

- Spectral only: It makes the analysis simpler and faster while providing accurate results when crosstalk is absent (and often even when it is present). This map does not include NPJ and BUJ categories.



Jitter map - Spectral Only

- Spectral + BUJ: This decomposition method includes an additional model component known as Non-Periodic Jitter (NPJ). This jitter model is more accurate when certain types of crosstalk are present. The disadvantage of using this map is that more statistics (a higher population of unit intervals) must be acquired before results can be produced. This may require longer record length, multiple acquisitions, or both.



Jitter map - Spectral + BUJ

Breakdown of noise (Noise Map)

The breakdown of noise into components such as Random Noise (RN) and Deterministic Noise (DN) is model-based. This means that a suitable mathematical model is proposed for the overall noise measurements, consisting of various components. The components are separable from each other based on observable characteristics, and the rules by which these components combine to form an overall noise distribution are based on well-understood mathematical principles.

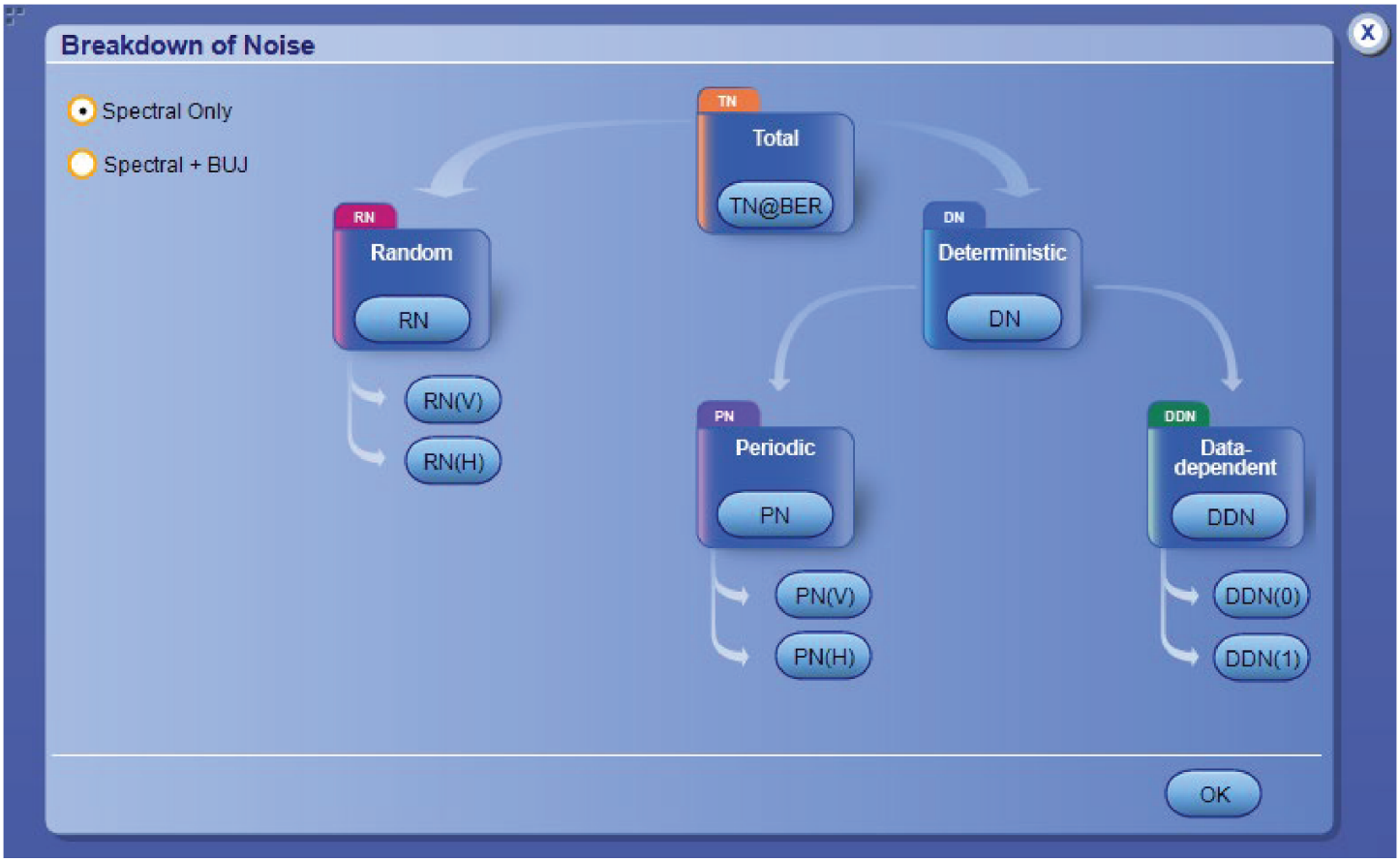
The model is hierarchical, and is represented by a noise map. Depending on the selected Jitter Separation Model, two distinct jitter maps are utilized. To select the Jitter Separation Model, follow the steps:

1. Tap Utility > User Preferences > Jitter and Eye Analysis.
2. Select the Jitter Separation Model as Spectral Only or Spectral + BUJ.

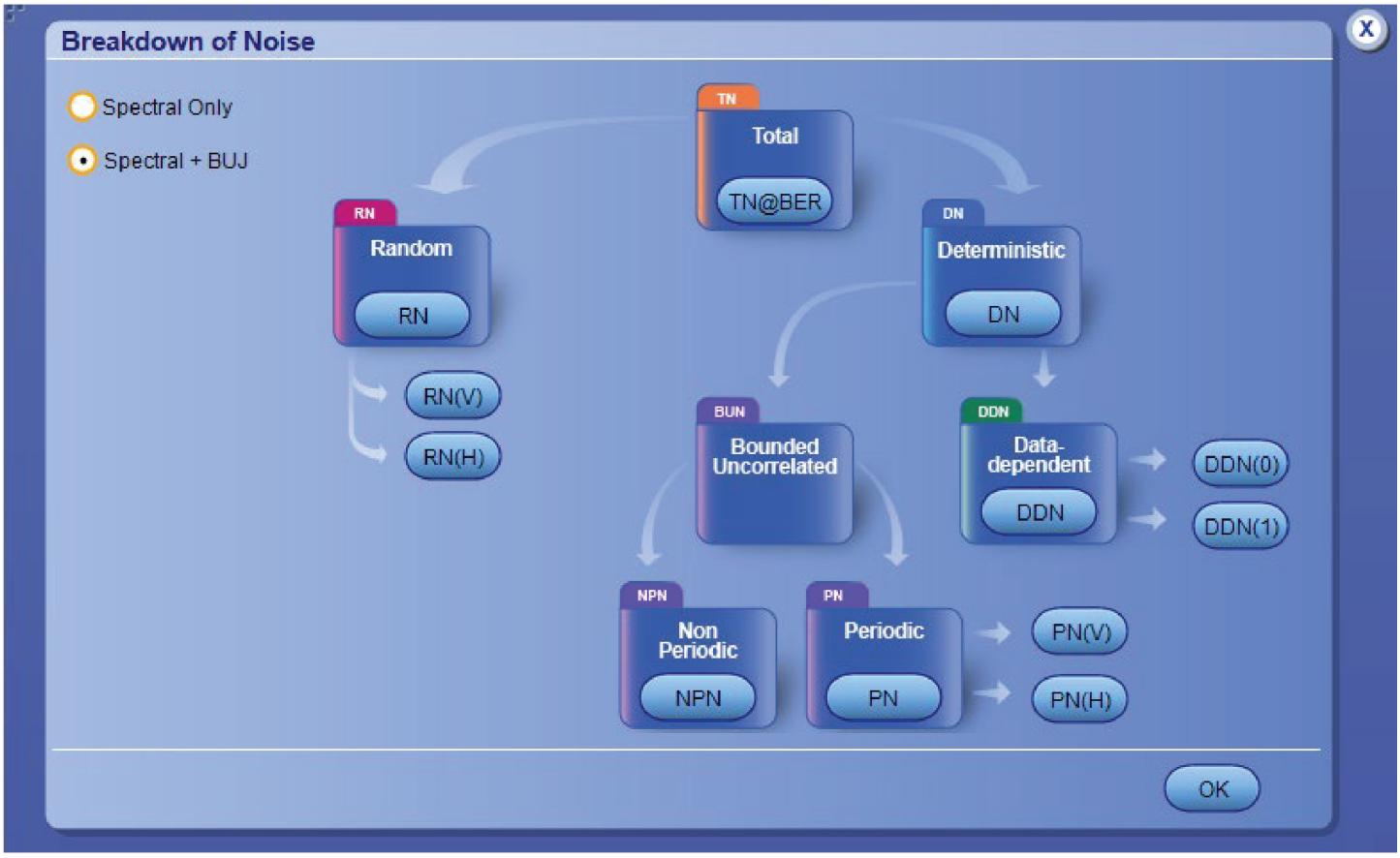
The two different jitter maps depending on the global configuration:

- Spectral only (default): A simpler map that does not include NPN and BUN categories is used when the decomposition method is set to 'Spectral only'. This offers simpler and faster processing and gives accurate results when crosstalk is not present (and often even when it is present).
- Spectral + BUJ: This decomposition method offers an additional model component (Non-Periodic Noise or NPN). This noise model is more accurate when certain types of crosstalk are present. The disadvantage of using this method is that more statistics (that is, a higher population of unit intervals) must be acquired before results can be produced. This may require longer record length, multiple acquisitions, or both.

By clicking the radio buttons in the upper left corner of the map window, you can switch between the two decomposition models and their corresponding maps. Once a model has been selected, you can add noise measurements by clicking directly on the buttons embedded in the map, or dismiss the map and click on the conventional buttons in the main control window.



Noise map - Spectral Only



Noise map - Spectral + BUJ

Jitter separation

Jitter analysis through RJ-DJ separation

Many of the jitter measurements are based on the concept of RJ-DJ separation. The application begins with the measured jitter-versus-time (as represented by the TIE measurement array) and analytically determines the random and deterministic components of the jitter. The deterministic part is further separated into independent subcomponents with specific characteristics.

The random jitter (RJ) is assumed to be zero-mean Gaussian, and is assumed to have a flat spectrum when viewed in the frequency domain. The measured RJ is fitted to a Gaussian mathematical model, which is parameterized by its standard deviation. Using the mathematical model for RJ, statistically probable jitter extremes may be predicted for much greater populations than actually measured.

The deterministic jitter (DJ) is predictable and can be generated consistently given known circumstances. The various DJ measurements each report the peak-to-peak value of the corresponding DJ subcomponent.

Once all the jitter components have been identified and the random jitter has been converted to a mathematical model, the components can be reassembled such that performance may be extrapolated to extremely low bit error rates. The probabilistic Total Jitter (TJ@BER) and probabilistic Eye Width (Width@BER) are examples of such measurements. The reported values are predictions that correspond to a user-specified Bit Error Rate, rather than observed values.

Two approaches are supported for performing jitter separation. The first method is based on spectrum analysis. It is only possible when the data pattern is repetitive. A clock waveform is always repetitive. Other repetitive testing data patterns are used, such as the K28.5 data pattern. Patterns may have rather long repetition lengths; for example, the CJTPAT pattern is 2640 bits. When using this method, you must specify the pattern length, and you will receive a warning if the pattern length appears to differ from that specified.

The second RJ-DJ separation method, known as arbitrary pattern analysis, may be used when the data pattern is not necessarily repetitive. This method works by correlating deterministic jitter observed over many repetitions with the bit pattern within a time-domain window surrounding each observation.

RJ-DJ separation via spectrum analysis

When the source waveform represents a repeating data pattern, Deterministic Jitter (DJ) has a frequency spectrum of impulses. The impulses due to the data pattern are equally spaced and occur at predictable frequencies related to the pattern length and bit rate. Specifically, the pattern-related jitter impulse must occur at multiples of f_o/N, where f_o is the data bit rate and N is the data pattern length. Other spectral impulses may occur due to periodic jitter not correlated with the data pattern.

To obtain measurements of DJ and RJ, all the components of the jitter spectrum that exceed the noise floor by a chosen margin are attributed to deterministic jitter. Those components that fall at the frequency increment corresponding to the pattern length are identified as data-dependent jitter, and those occurring at other frequencies are attributed to uncorrelated periodic jitter. The remaining spectral noise floor (appropriately normalized to account for the removed deterministic jitter) is integrated to predict the standard deviation of the underlying Gaussian random noise process.

Once the spectral components corresponding to each deterministic jitter type have been identified, each component is inverse-transformed back to the time domain. From these waveforms, the peak-to-peak jitter for each component is determined. For the random jitter, the RMS deviation is directly computable from the standard deviation of the Gaussian model.

RJ-DJ separation for arbitrary patterns

When the data pattern borne by the source waveform is not cyclically repeating, any periodic jitter still has a frequency spectrum consisting of impulses but this is not true of the data-dependent jitter.

In this case, analysis of the data-dependent jitter may proceed based on the assumption that any given bit is affected by a finite (and relatively small) number of preceding bits. By averaging all events for which a given bit is preceded by a particular bit sequence, the data-dependent jitter attributable to that bit sequence is obtained. This is because PJ and RJ are not correlated to a particular data sequence and thus are averaged out.

If each bit is assumed to be affected by N preceding bits, there are a total of 2N possible data sequences. The sequence length N is a configurable parameter. To get statistically sound average values for the data-dependent jitter, a minimum population of observations is required for each individual pattern that occurs at least once. This population limit is also configurable by the user.

By the above means, the data-dependent jitter is characterized. Once characterized, the data-dependent jitter, on a bit-by-bit basis, may be removed from the original jitter versus time record. The remaining jitter is composed of periodic and random jitter. This jitter is transformed into the frequency domain, and the spectral analysis approach is used to separate the impulsive periodic jitter from the broad noise floor of random jitter.

Separation of Non-Periodic jitter (NPJ)

Bounded Uncorrelated Jitter (BUJ) refers to all bounded jitter that is not correlated to the data pattern on the waveform. Thus, it excludes DDJ, DCD and RJ. It can be further subdivided into PJ (which is deterministic, and easily recognized using a spectral approach) and Non-Period Jitter (NPJ).

Depending on its precise nature, NPJ is often difficult to distinguish from RJ. It typically cannnot be isolated using frequency domain techniques, so a different domain is used. The important difference between RJ and NPJ is that RJ has a Gaussian distribution (with unbounded tails) whereas NPJ is bounded by definition. This fact is used as a basis for separation.

The jitter separation algorithms are modified as follows when the Spectral + BUJ method is selected:

1. Data-Dependent Jitter (DDJ) is first removed using either a spectral approach for repeating patterns or a correlation approach for arbitrary data patterns.
2. PJ is identified and removed using a spectral approach.
3. The remaining jitter is assumed to contain (Gaussian) RJ and possibly NPJ. (In the Spectral Only method of jitter analysis, no further processing is done and this jitter is reported as RJ.)
4. When Spectral + BUJ processing is selected, the RJ + NPJ jitter is collected into a histogram that is typically accumulated over multiple waveform acquisitions. When the (user-configurable) minimum histogram population has been acquired, the histogram is converted to an estimate of the Cumulative Density Function (CDF) and plotted using the Q-scale for the vertical axis. The Q-scale plot has the property that a true Gaussian distribution appears as a straight line, with a slope equal to 1/σ (where σ is the standard deviation of the Gaussian distribution). If the distribution is a mixture of a true Gaussian plus some bounded distribution, the plotted curve has left and right extremes that asymptotically approach straight lines with a slope of 1/σ. The horizontal offset between the two asymptotes represents the dual-dirac magnitude of the NPJ.

Estimation of TJ@BER and eye Width@BER

One of the outcomes of the RJ-DJ separation was a mathematical model for random jitter's probability density function (PDF) and measured values for the PDFs of the deterministic jitter components. Since all of these components are assumed to be statistically independent, the PDF of the total jitter can be calculated by convolution.

Integration of the PDF yields the cumulative distribution function (CDF), which can then be used to create the bit error rate curve (bathtub curve). Based on the bathtub curve, the eye opening (Width@BER) and eye closure (TJ@BER) can be estimated for a given bit error rate.

The application calculates the eye opening at the specified BER using the following equation:

Eye opening = 1–TJ@BER when TJ@BER is less than one Unit Interval

Jitter estimation using Dual-Dirac models

Jitter estimation based on RJ-DJ separation depends in part on the specific jitter components modeled. For the purposes of analyzing jitter and identifying root cause, it is very useful to identify components as specifically as possible. But for the purposes of determining compliance, it has been found that a simplified jitter model yields results that are more consistent across different measurement instruments and different vendors.

A simplified model that has found acceptance in several industry standards is known as the Dual-Dirac model. This is because the probability density function (PDF) of all the deterministic jitter is replaced with a PDF consisting of two Dirac functions such that the total jitter and eye opening at very low bit error rates is unchanged. The Random Jitter and Deterministic Jitter values derived from this model are identified as RJ–δδ and DJ–δδ, respectively.

Joint Jitter/Noise analysis

Differences between Jitter-Only and Jitter+Noise analysis

Traditionally, jitter has been analyzed by detecting edge crossings at a voltage, current power reference level and then analyzing the distribution of these edge times relative to a reference clock. Based on various properties of the jitter distribution (observed in the time, frequency and statistical domains), the jitter has been separated into categories or components that exhibit well-known behaviors. This allows a mathematical model of the jitter to be constructed, which can then be used to extrapolate jitter behavior for higher populations of bits than were actually measured. In this process, the shape of the analog waveform between edge crossings is entirely ignored.

In Joint Jitter/Noise Analysis, the entire analog waveform is analyzed. The process is analogous in many ways to Jitter-Only analysis, but the mathematical model for jitter and noise behavior is more thorough and better fits the physical world. For example, in addition to the familiar jitter components (RJ, PJ, DDJ) there are corresponding noise components (RN, PN, DDN). Jitter is still analyzed at points where the waveform crosses a horizontal reference line, and Noise is analyzed where the waveform crosses a vertical reference line once each unit interval. These two reference planes are where the model parameters (RJ – DDJ and RN – DDN) are determined, although the process of determining the parameter values involves compensating for effects of jitter on noise and vice versa.

Because both techniques use parameterized mathematical models to extrapolate behavior, they will not yield identical results (but should be very close). Consider an analogy: You could measure the area of a circle by splitting the circle into a number of narrow wedge-shaped triangles with their points all at the center of the circle, and adding the areas of the triangles. Or you could estimate the circle's area by placing a fine rectangular grid on the circle and counting the number of squares that were mostly inside the circle. Both methods will yield a good estimate but the two models won't give the same answer.

Because the Jitter-Only model does not consider vertical effects, there are some aspects of eye closure to which the jitter-Only method is entirely blind but the Jitter+Noise method models well. The Jitter-Only method does not provide any noise measurement values at all, for obvious reasons. And the Jitter+Noise method exposes how noise is manifested as jitter, and jitter as noise, which is helpful in identifying root cause.

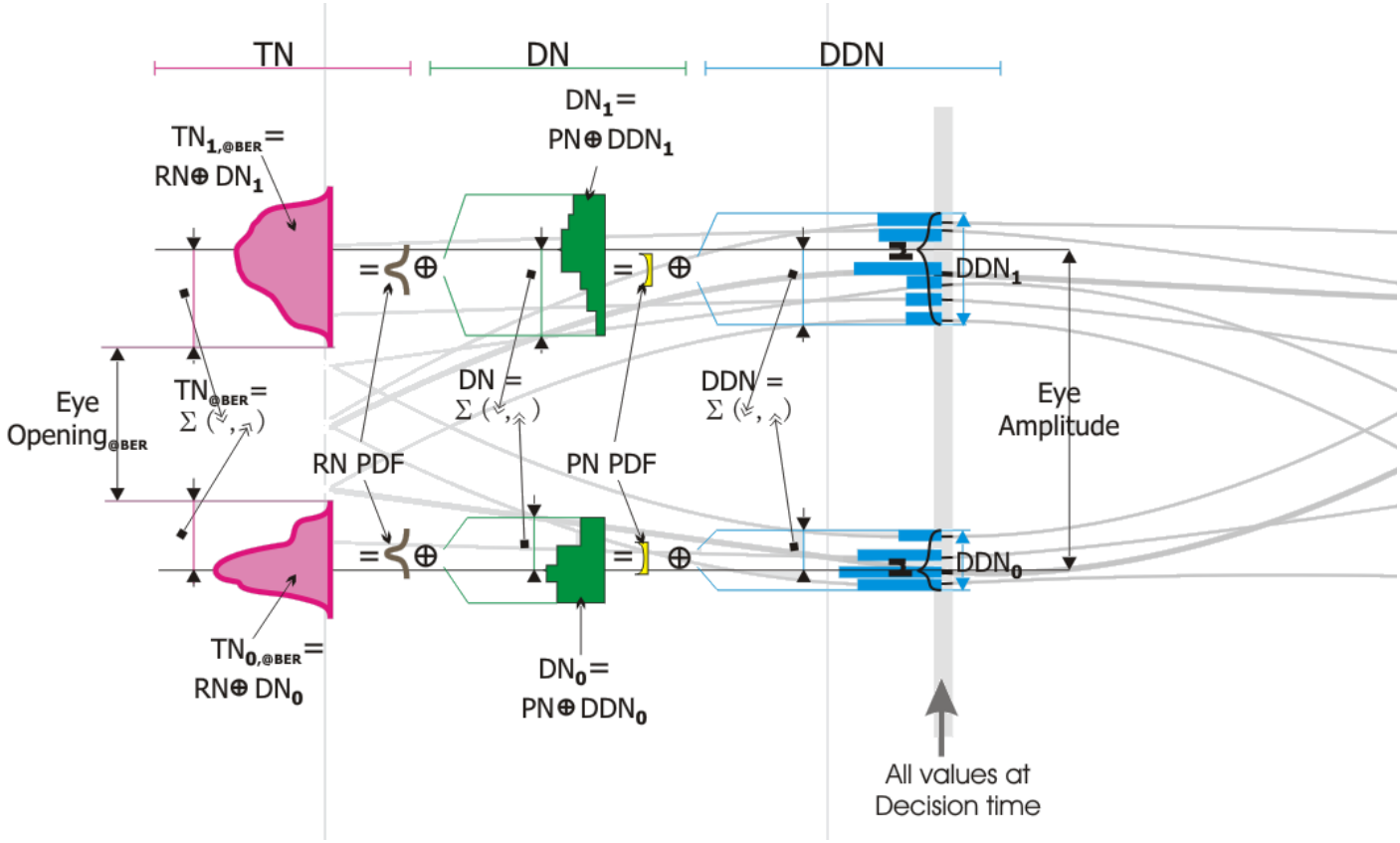
Basic steps in joint Jitter+Noise analysis

The process of performing Jitter+Noise analysis includes these major steps:

1. Perform clock recovery to identify a reference clock representing the ideal transition times. This process also yields the received Boolean bit stream.
2. Measure the exact instant in time where each data transition (edge) occurs.
3. Measure the slew rate at each of these edges.
4. Measure the exact amplitude at the center (50% point) of each bit.
5. Measure the slew rate at each of these bit centers.
6. Perform a joint analysis on the prior four data sets to decouple the effects of jitter on noise and vice versa.
7. Considering the entire analog waveform (not just the edges and bit centers), derive the "correlated waveform". This is the waveform the reflects channel effects (low-pass filtering, reflections) while filtering out variations uncorrelated to the data pattern.
8. Compare the edge times from step 2 with the clock times from step 1 to derive TIE (Time Interval Error). Use spectral analysis, time analysis and Q-scale analysis to derive model parameters such as RJ and PJ.
9. Compare the bit amplitudes from step 3 with the ideal bit stream (derived from the received Boolean bit stream and the nominal high and low bit amplitudes). Use spectral analysis, time analysis and Q-scale analysis to derive model parameters such as RN and PN.
10. Use two-dimensional convolution to combine the effects of all jitter an noise model parameters into a statistical model of uncorrelated noise. This includes RJ, PJ, NPJ, RN, PN and NPN.
11. Convolve the model of uncorrelated jitter+noise with the correlated waveform to create a model of the overall eye diagram's true PDF.
12. Integrate the PDF of step 11 from top to bottom (for '0' bits only) to derive the '0' bit CDF. Perform a complimentary operation to derive the '1' bit CDF.
13. Superimpose the CDFs of steps 11 and 12 to get the BER eye: analogous to the bathtub curve, except two-dimensional. (In fact, the jitter bathtub or the noise bathtub can be derived directly from the BER Eye by slicing it either horizontally or vertically, respectively.)

Noise model component interrelationships

The figure below depicts graphically how the various noise model parameters relate to each other. It is only the upward-reaching portions of the lower models and the downward-reaching portions of the upper models that contribute to eye opening (or closure), and this is reflected in how the DDN, DN and TN@BER values are calculated.



Noise model component interrelationships

Timing measurement algorithms

Burst Width measurement algorithm

Burst Width is the duration of a series of adjacent crossings of the mid reference level. The duration of a burst. Bursts are separated by a user-defined idle time.

Data Rate measurement algorithm

Data Rate is the reciprocal of Unit Interval. This measurement is made on each bit in the record.

Delay measurement algorithm

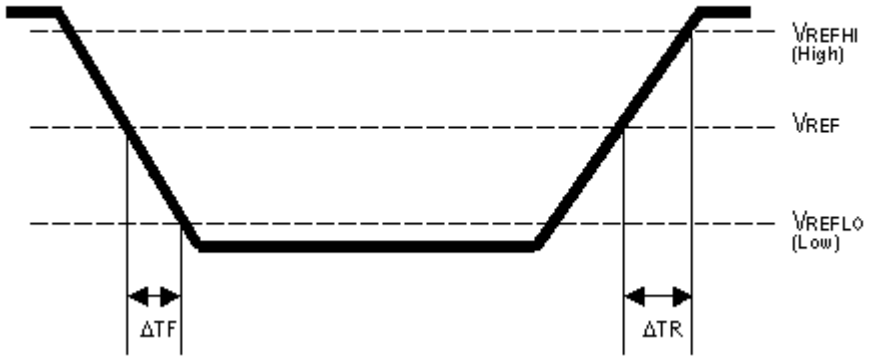
Delay is the time between a mid reference level edge on one source to a mid reference level edge on a second source. The direction of each edge can be configured by the user.

Falling slew rate

Falling Slew Rate is the rate of change in value as an edge transitions from the high or mid reference level to the mid or low reference level. The levels are configurable.

In the diagram below, the Falling Slew Rate from mid ref to low ref is calculated using the following equation:

Falling Slew Rate = (VREF - VREFLO)/ΔTF

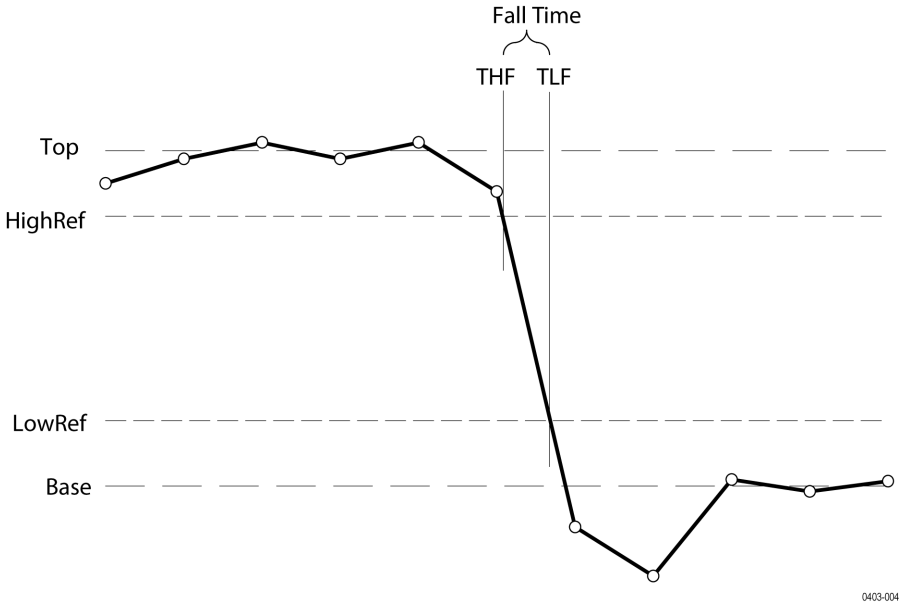


Fall Time measurement algorithm

Fall Time is the time required for an edge to fall from the high reference level to the low reference level. By default the measurement is from reference level 90% amplitude to 10% amplitude.

The following figure shows a falling edge with the two edges necessary to calculate a Fall measurement. The figure shows the default high reference level which is 90% of Top and the default low reference level which is 10% of Base.

1. Searching from *Start* to *End*, find the first sample in the measurement zone greater than *HighRef*.
2. From this sample, continue the search to find the first (negative) crossing of edge of *HighRef*. The time of this edge is *THF*. (Use interpolation if necessary.)



Fall Time

1. 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.)
2. *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.

T Clock is the VRefMid crossing time for the configured edge direction.

If the Signal Type is Data

The N–Period measurement calculates the elapsed time for N consecutive unit intervals.

The application calculates this measurement using the following equation:

$$NP_n^{Data} = T_{n+N}^{Data} - T_n^{Data}$$

Where:

NPData is the duration for N unit intervals.

TData is the VRefMid crossing time in either direction.

If *Tn+NData* does not exist for a given n, no measurement is recorded for that position.

Negative Duty Cycle measurement algorithm

Negative Duty Cycle is the ratio of the negative pulse width to the signal period expressed as a percentage.

NegativeWidth is defined in *Negative Pulse Width*, below.

If *Period* = 0 or undefined then return an error.

$$NegativeDutyCycle = \frac{NegativeWidth}{Period} \times 100\%$$

0020-018

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

The time between two adjacent crossings of the Mid reference level (one cycle) of the waveform. Period is measured in horizontal units, typically seconds.

Period = Edge3 - Edge1

Phase measurement algorithm

Phase is the ratio of the Skew between two sources to the Period of the first source. It is the amount of phase shift between edges of the two waveforms. The phase shift is expressed in degrees of the Source1 waveform cycle, where 360 degrees is one complete cycle (Period) of Source1. For best results, Source1 and Source2 should be of the same frequency or one waveform should be a harmonic of the other.

Phase is determined in the following manner for each cycle of the record:

1. The first two adjacent edges of the configured 'from' edge type are found in Source1.
2. The period of Source1 is calculated (see *Period* above). If the "from" edge is set to either, the half-period of Source1 is calculated.
3. The first edge in Source2 in the configured 'to' edge direction is found.
4. The Skew from the first Source1 edge to the next Source2 edge is calculated.

   (Skew/Period)*360
5. When "from" edge is either, the calculation is (Skew/half-Period)*180.

Positive Duty Cycle measurement algorithm

Positive Duty Cycle is the ratio of the positive pulse width to the signal period, expressed as a percentage.

PositiveWidth is defined in *Positive Pulse Width*, following.

$$PositiveDutyCycle = \frac{PositiveWidth}{Period} \times 100\%$$

0020-021

Positive Pulse Width measurement algorithm

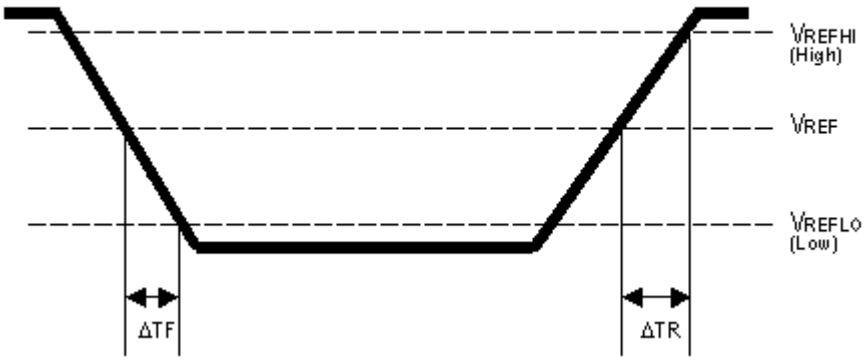
Positive Pulse Width is the time the signal remains above the mid reference level. It is the distance from a rising edge to the next falling edge.

Rising Slew Rate measurement algorithm

Rising Slew Rate is the rate of change in value as an edge transitions from the low or mid reference level to the mid or high reference level. The levels are configurable.

In the diagram below, the Rising Slew Rate from mid ref to high ref is calculated using the following equation:

(VREFHI - VREF)/ΔTR

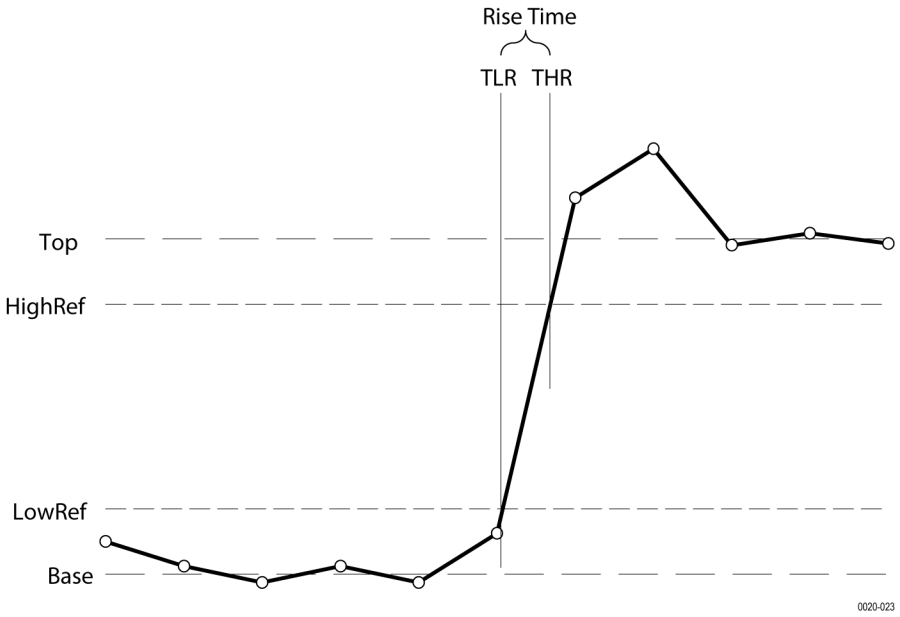


Rise Time measurement algorithm

Rise Time is the time required for an edge to rise from the low reference level to the high reference level. By default the measurement is from reference level 10% amplitude to 90% amplitude.

The following figure shows a rising edge with the two crossings necessary to calculate a Rise Time measurement.

1. Searching from *Start* to *End*, find the first sample in the measurement zone less than *LowRef*.
2. From this sample, continue the search to find the first (positive) crossing of *LowRef*. The time of this crossing is the low rise time or *TLR*. (Use linear or sin interpolation if necessary.)
3. From *TLR*, continue the search, looking for a crossing of *HighRef*. Update *TLR* if subsequent *LowRef* crossings are found. If a *HighRef* crossing is found, it becomes the high rise time or *THR*. (Use linear or sin interpolation if necessary.)
4. *RiseTime = THR – TLR*



Setup

Setup Time is the time between the mid reference level crossing of the clock source (Source1) and the closest previous mid reference level crossing of the data source (Source2). The crossings (edges) may be configured to be rising, falling or either.

The application calculates this measurement using the following equation:

$$T_n^{Setup} = T_i^{Main} - T_n^{2nd}$$

Where:

T Setup is the setup time.

T Main is the Main input (clock) Mid reference crossing time in the specified direction.

T 2nd is the 2nd input (data) Mid2 reference crossing time in the specified direction.

Skew

Skew is the time between the mid reference level crossing on Source1 to the mid reference level crossing on Source2. The direction of the edge crossing is configurable.

The application calculates this measurement using the following equation:

$$T_n^{Skew} = T_n^{Main} - T_n^{2nd}$$

Where:

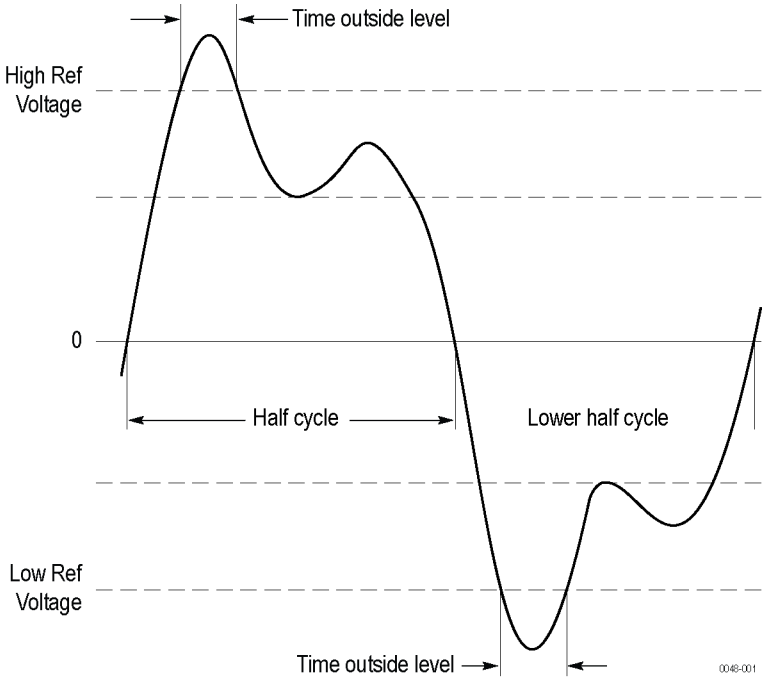
T Skew is the timing skew.

T Main is the Main input Mid reference crossing time in the configured direction.

T 2nd is the 2nd input Mid2 reference crossing time in the configured direction.

Time Outside Level measurement algorithm

Time Outside Level is the time the signal remains above the high reference level and/or below the low reference level.



Time to Max

Time to Max is the amount of time from the trigger point to the maximum data point. This measurement can be made across the entire record or on each cycle in the record.

Time to Min

Time to Min is the amount of time from the trigger point to the minimum data point. This measurement can be made across the entire record or on each cycle in the record.

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.

TDR measurement algorithm

TDR computes the impedance waveform based on reflections from incident step function wave traveling through transmission media.

The voltage waveform measurement is transformed into reflection coefficient (ρ) using the following equation:

$$\rho[k] = \frac{V[k] - V_{mean}}{V_{amp}} - 1$$

Where:

V_{mean} is mean of complete voltage waveform

V_{amp} is amplitude of incident step voltage waveform

Index *k* indicates one measurement per sample.

Impedance (Z) waveform is derived from Rho waveform using the following equation:

$$Z[k] = Z_0 * \frac{(1 + \rho[k])}{(1 - \rho[k])}$$

Where:

Z_0 is Intrinsic impedance of TDR source and oscilloscope. Default value is 50 Ω.

Index *k* indicates one measurement per sample.

If the Signal Type measurement configuration is set to Differential, then each line of TDR is converted to Z waveform using the equations mentioned above. Rho waveform of each line is always represented from -1ρ to 1ρ.

Performing differential TDR with identical SMA cables ensures that both positive and negative TDR voltages are deskewed.

Differential Z waveform is computed by adding both lines of impedance waveforms using the following equation:

$$Z_{diff}[k] = Z_{TDR}[k] + Z_{\sim TDR}[k]$$

Where:

Z_{TDR} is impedance wave derived from the positive TDR Voltage waveform.

$Z_{\sim TDR}$ is impedance wave derived from the negative TDR Voltage waveform.

Index *k* indicates one measurement per sample.

Mean impedance value in the measurement region of Z waveform is displayed on badge using the following equation:

In all the below equations, Start and End defines the measurement region points using the measurement gating option.

$$Zmean = \int_{Start}^{End} \frac{Z(t)dt}{((End - Start) * SampleInterval)}$$

Where, *Z(t)* is Waveform (Z) at time (t).

Capacitance or inductance value at measurement region of Z waveform is displayed on the badge using the following equations:

$$\text{Cmean} = \frac{2.0}{Z_0} * \int_{Start}^{End} \frac{\rho(t)dt}{((\text{End} - \text{Start}) * \text{SampleInterval})}$$

$$\text{Lmean} = (2.0 * Z_0) * \int_{Start}^{End} \frac{\rho(t)dt}{((\text{End} - \text{Start}) * \text{SampleInterval})}$$

Where:

Z_0 is Intrinsic impedance of TDR source and oscilloscope. Default value is 50 Ω.

Index *t* indicates that it is one measurement per sample.

NOTE: If the incident step is not determined during acquisition, then the Zmean, Cmean, and Lmean algorithms assume that the incident step is located at time zero.

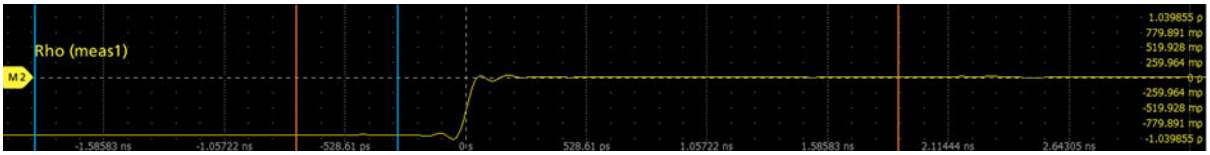
If the algorithm is not able to determine the reflection and the gating type is set to None, then the Zmean, Cmean, and Lmean algorithms assume that the reflection is located at time 100 ns.

TDR preset algorithm

TDR Preset will set up trigger to negative TDR source and sets up channel with best bandwidth settings along termination as 50 Ω. TDR Preset calibration is performed in average acquisition mode and optimal sample rate.

Reflection coefficient (ρ) waveform is derived out of input voltage waveform. TDR Preset will calibrate short condition using mean of ρ waveform between region defined by blue vertical lines before incident, refer to the "Rho calibration regions" figure below. The calibration constants are adjusted such that mean of ρ waveform in the region is matched with -1ρ level.

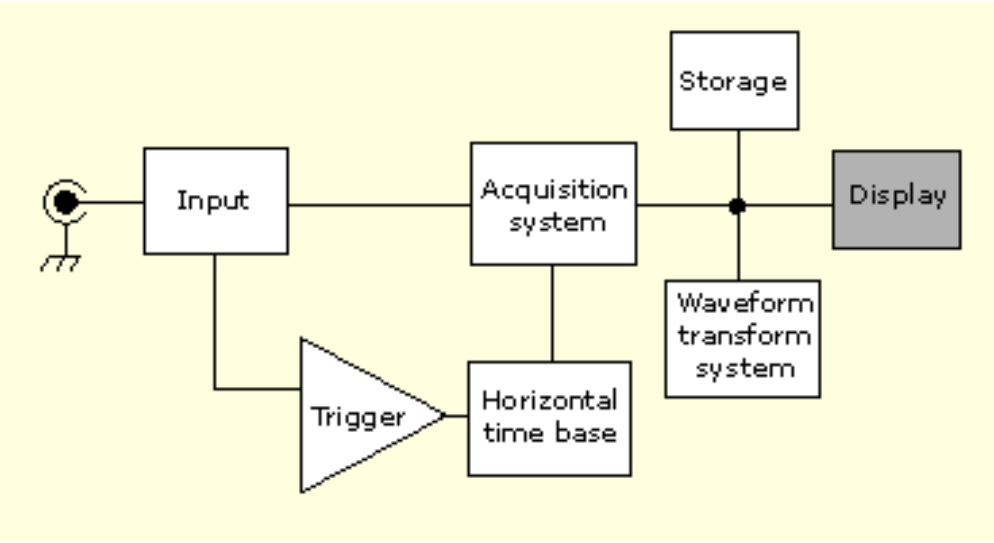
Once the short is calibrated, matched impedance condition is calibrated by determining amplitude between region defined by orange vertical lines, refer to the "Rho calibration regions" figure below. The calibration constants are adjusted such that amplitude of ρ waveform in the region is matched with 1ρ. This adjustment makes sure matched impedance is at 0ρ.



Rho calibration regions

Waveform display concepts

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



The display shows analog, 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).

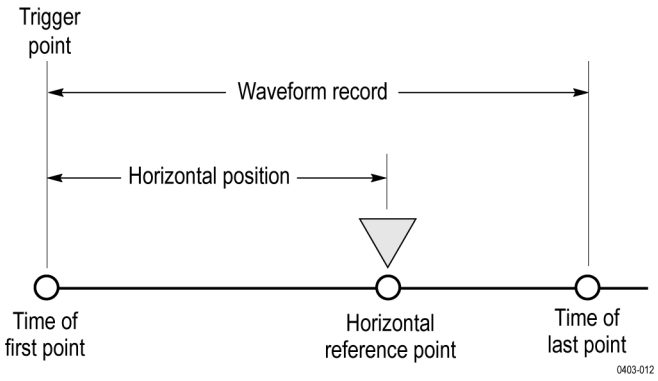
You can also add histogram, eye diagram, and measurement results views (plots) for individual measurements. These plot views are contained within their own view window and can be moved within the overall view area.

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.



What do you want to do next?

Learn about using the MultiView Zoom feature. (on page 58)

Go to a step-by-step procedure for displaying waveforms. (on page 89)

References

SIM badge error and warning messages

The following table provides information to help resolve error or warning messages that appear on the SIM badge.

Error	Cause	Suggestion
Scope SR not set to <SR>	To ensure correct operation of the SIM model, the oscilloscope sample rate must be configured to match the sample rate used in model simulation. If there is a mismatch, an error badge appears. When this condition occurs, all test point math waveforms disappear (waveform visuals only, while the waveform handles and associated badges remain visible).	Match the oscilloscope sample rate to the model's sample rate or vice versa and recalculate the model.
Model Recalc Required	Any changes to the vertical settings associated with one or more of the model's inputs that require new S-parameters to be passed to the model will cause this error to be shown in the SIM badge. In this case, all test point math waveforms disappear (waveform visuals only, while the waveform handles and associated badges remain visible).	Open the SIM model and recalculate it with the new S-parameters or revert the vertical settings to previous configuration.
Model Requires >= <RL>	This error can occur in two scenarios. If the model does not include a receiver, and the user modifies the oscilloscope settings such that the RL drops below 50k, an error appears in the SIM badge. Or if the model includes a receiver block, and the user modifies the oscilloscope settings such that the RL drops below 400k, then an error appears in the SIM badge. In both the cases, the oscilloscope continues to display test point math waveforms, but the badge displays an alert indicating the results being displayed on screen do not match the simulation definition.	Increase the record length to the value mentioned in the error message.
Probe cannot be set to CM	If a TriMode probe is in use and the user changes the probe operating state to Common Mode, then this error appears in the SIM badge.	Set the probe mode to either Differential, A, or B mode.

Error	Cause	Suggestion
Error in test-point co-efficient computation.	If there is a failure in the test point co-efficient computation, then this error appears. Incorrect port assignments in a Block. Excessive ratio between Source Sample Rate and Frequency Spacing in Block models (including S-parameters). Consider increasing Frequency Spacing. Too many Blocks in the configuration. SIM calculation stops when the algorithm detects model issues.	Review your Block settings, correct any errors, and re-run Calculate SIM.
S-parameters file content does not match the specified file format (<.sXp>) for Block: <block label>	If there is a mismatch between the S-parameters file name extension and the content, then this error appears. For example, extension is s4p but the contents are s2p.	Go to the block mentioned in the error message and replace the S-parameters with the right content.
S-parameters file is invalid for Block: <block label>	If the read operation of a S-parameters file failed, then this error appears.	Refer to the next error in the pop-up menu.
Unable to determine symbol rate.	Clock recovery is unable to detect the symbol rate for various reasons.	Set the **Symbol Rate Detect** to **Manual** and enter the Nominal Symbol Rate.
Signal to noise ratio not high enough for clock recovery.	Clock recovery did not process the waveform due to low signal to noise ratio.	Set the **Symbol Rate Detect** to **Manual** and enter the Nominal Symbol Rate. The input waveform can be filtered using Rx filter/Global Bandwidth/CTLE to reduce the noise.
Not enough edges to recover clock.	The input waveform does not have adequate number of edges required for the clock recovery.	Increase record length or PLL loop bandwidth.
Clock is too unstable.	Recovered clock has way too much noise.	Verify symbol rate in Clock Data Recovery or DUT.
Symbol rate too close to Nyquist frequency. Increase sampling rate or reduce symbol rate.	Symbol rate is close to half of the sample rate.	Increase sample rate or decrease symbol rate.
DFE / FFE adaptation failed. Recommend checking symbol rate and clock recovery settings.	DFE/FFE adaptation failed due to high RMSE error or noise in the input waveform.	Verify symbol rate in Clock Data Recovery or DUT blocks. The input waveform can be filtered using Rx filter/Global Bandwidth/CTLE to reduce the noise.

TDR badge error and warning messages

The following table lists the errors and warning messages on the TDR measurement badge.

Error messages displayed on TDR measurement badges

Error Message	Cause	Suggestion
Empty input	The oscilloscope is waiting for a single sequence trigger. Input waveform has no data.	Check the TDR source connections. Check that the oscilloscope has a valid input waveform and valid trigger.
Too few edges	Number of edges found in the gated region is less than the number of edges required for the algorithm.	Increase the record length. Change the gating region.
Invalid input	Input waveform has invalid samples.	Re-acquire the input waveform.
Input source mismatch	In the case of two Ref waveforms, the two waveforms recalled must have the same record length and have been acquired with the same time base settings.	Acquire two waveforms of the same type (active or ref). Re-acquire the ref waveforms with the same time base and record length.
Unable to get active TDR source list	Could not turn on display of voltage source set for TDR measurement.	Manually turn on the channel. Change the voltage source.
Oscilloscope Autoset is failing. Check connections and return TDR Preset.	Autoset is intermediate step of TDR preset. It failed due to invalid voltage source provided for the TDR measurement.	Check the TDR connections. Verify the TDR measurement settings. With valid voltage source re-run the TDR Preset.
Horizontal settings failed	When TDR Preset could not set valid sample rate, horizontal scale, or acquisition mode.	Perform the TDR Preset again. If issue persists restart the device.
Vertical settings failed	When TDR Preset could not set coupling, termination, and bandwidth.	Perform the TDR Preset again. If issue persists restart the device.
Unable to set oscilloscope to run mode	When TDR Preset could not set the oscilloscope to free mode.	Perform the TDR Preset again. If issue persists restart the device.
Input signals are not connected. Power on the DUT and connect the input signals to configure channels.	TDR Preset could not find valid signal.	Check the TDR connections and measurement source. Perform the TDR Preset again.

Warning messages displayed on the TDR measurement badges

Warning Message	Cause	Suggestion
Incident step not found	Could not detect falling edge of incident step for negative TDR voltage waveform or could not detect rising edge for positive TDR voltage waveform.	Check the TDR source connections. Check the TDR measurement voltage source. While displaying the warning, incident step is assumed at zero second. Ignore this warning when signal type is differential and DUT is matched with characteristic impedance.
Reflection not found	Could not detect rising or falling edge on the TDR voltage waveform after incident step. Could not automatically detect region of interest to determine impedance.	This is informative warning for user. No action required if DUT is matched with characteristic impedance.

Warning Message	Cause	Suggestion
Deskew input source	Only for the differential TDR when negative TDR voltage waveform exhibits a skew with respect to positive TDR voltage.	In any one of the differential TDR voltage waveforms, change the skew settings of the voltage waveform.
Improper settings	Indicates when the TDR measurement requires calibration. This could be due to the TDR setup issues, calibration issues, or oscilloscope settings issues.	Check the TDR setup connections. Perform the TDR Preset. For seldom warning during oscilloscope acquisition in average mode, ignore the warning.

User defined filter errors and warnings

The following table lists the errors and warning messages for the user defined filters.

Message	Description
Info: To generate filter response, click on Apply Filter	The Filter response will be blank by default. Run filter to generate the response.
Info: Filter is valid for these sample rates 1 MS/s - 12.5 GS/s	Filter is designed for cutoff frequencies in the range of 0.05 to 0.45 times the Oscilloscope Sample Rate. This message informs the user on the range of valid sample rates for which the filter can be designed.
Error: Odd order is recommended for High-Pass Gaussian filter Error: Odd order is recommended for Band-Stop Gaussian filter	For Gaussian Filters, results are unreliable with the even filter order. So design the filter for Odd order to get a valid filter response.
Error: Filter settings could not be restored	When the loaded filter file is not created using UDF, filter settings cannot be restored and the message indicates that the filter file is not readable.
Error: Invalid SR. Filter is incompatible with the current sample rate	This message will display on the Math badge when there is a mismatch between sample rate configured in Oscilloscope horizontal time base and the sample rates for which filter is designed. To view the filtered signal, set the Oscilloscope sample rate to any of the valid filter sample rates.
Error: Insuf time. Filter requires longer acquisition time	This message will display on the Math badge when the horizontal time base is insufficient. Increase the horizontal scale to view the filtered signal on math.
Error: No valid sample rates for given filter settings	This message will display when the tool couldn't get valid oscilloscope sample rate for the requested cut off frequency/frequencies. Math Badge displays No Filter applied.
Error: Filter order cannot exceed 15k. Increase the transition width.	This message will display on filter creator window when the calculated order is greater than 15k. Increase the transition width to generate a valid response.
Error: Reduce sample rate to get the filter response	This message will display on filter creator window when the number of coefficients go beyond 64000. Reduce the sample rate to generate a valid response.